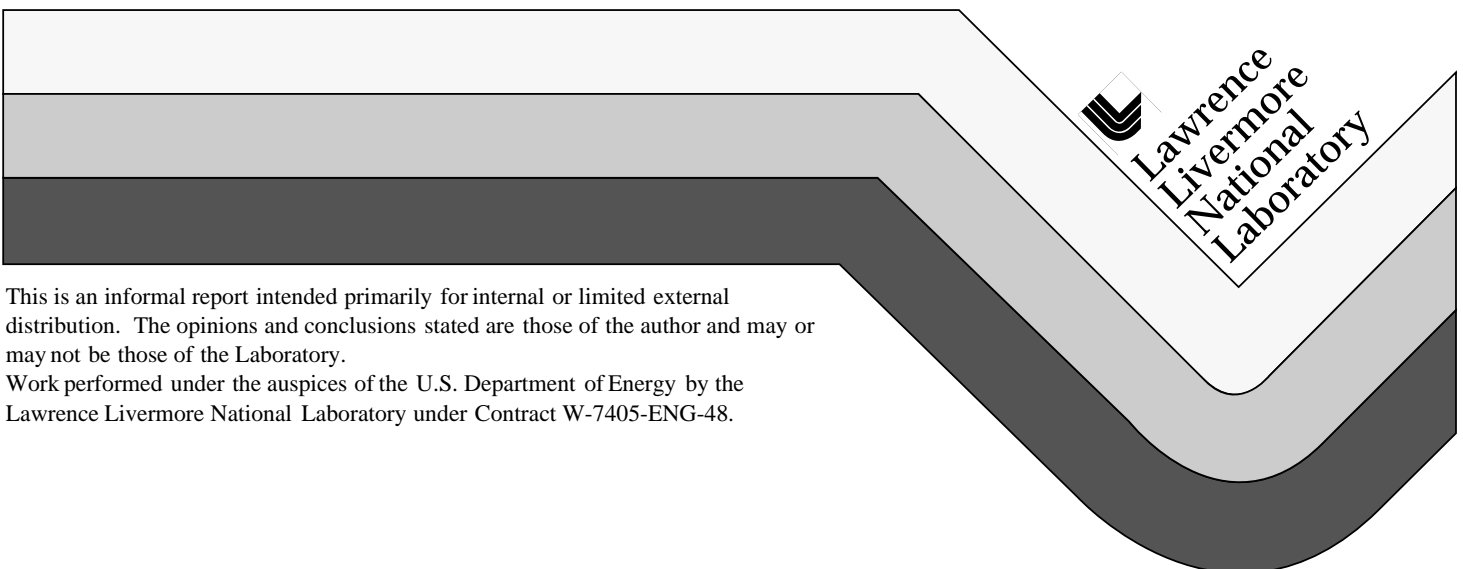


August 2000 Working Group Meeting on Heavy Vehicle Aerodynamic Drag: Presentations and Summary of Comments and Conclusions

Rose McCallen, Dan Flowers, Tim Dunn, Jerry Owens, Greg Laskowski , LLNL; Fred Browand and Mustapha Hammache, University of Southern California; Anthony Leoard and Mark Brady, California Institute of Technology; Kambiz Salari and Walter Rutledge, Sandia National Laboratories; James Ross, J.T. Heineck, Tom Arledge, Bruce Storms, NASA Ames Research Center

September 18, 2000



DISCLAIMER

This document was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor the University of California nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or the University of California, and shall not be used for advertising or product endorsement purposes.

This report has been reproduced
directly from the best available copy.

Available to DOE and DOE contractors from the
Office of Scientific and Technical Information
P.O. Box 62, Oak Ridge, TN 37831
Prices available from (615) 576-8401, FTS 626-8401

Available to the public from the
National Technical Information Service
U.S. Department of Commerce
5285 Port Royal Rd.,
Springfield, VA 22161

August 2000 Working Group Meeting on Heavy Vehicle Aerodynamic Drag: Presentations and Summary of Comments and Conclusions

Jointly written by
**Lawrence Livermore National Laboratory
Sandia National Laboratories
University of Southern California
California Institute of Technology
NASA Ames Research Center**

Introduction

A Working Group Meeting on Heavy Vehicle Aerodynamic Drag was held at Lawrence Livermore National Laboratory on August 16 - 17, 2000. The purpose of the meeting was to present technical details on the experimental and computational plans and approaches and provide an update on progress in the analysis of experimental results, model developments, simulations, and an investigation of an aerodynamic device. The focus of the meeting was a review of University of Southern California's (USC) experimental plans and results, NASA Ames experimental plans, the computational results from Lawrence Livermore National Laboratory (LLNL) and Sandia National Laboratories (SNL) for the integrated tractor-trailer benchmark geometry called the Ground Transportation System (GTS) Model, and turbulence model development and benchmark simulation for rounded cube shapes representative of a tractor and trailer being investigated by the California Institute of Technology (Caltech). Much of the meeting discussion involved wind tunnel testing plans, analysis of existing experimental data, simulation results, and needed modeling improvements. The present and projected budget and funding situation was also discussed.

Presentations were given by representatives from the Department of Energy (DOE) Office of Transportation Technology Office of Heavy Vehicle Technology (OHVT), LLNL, SNL, NASA Ames, USC, and Caltech. An industrial representative from International Truck and Engine Corporation participated in discussions and presented an industrial perspective. In addition, an overview of the laboratory was provided by an LLNL representative from their Engineering Directorate, and an update on the 21st Century Truck initiative was given by an LLNL representative from their Energy Directorate. This report contains the technical presentations (viewgraphs) delivered at the

Meeting, briefly summarizes the comments and conclusions, and outlines the future action items.

Summary of Major Issues

There were 3 major issues raised at the meeting.

1. With the projected funding for FY01, the desired experiments in the NASA 12' pressure wind tunnel (PWT) for the investigation of Reynolds number sensitivity can not be performed. Plans are to produce a multi-year plan to achieve this goal.
2. Another issue related to issue 1 is the choice of geometry for wind tunnel testing. The options are 1) Continue testing our Ground Transportation System (GTS), 2) Obtain a model from industry, or 3) Develop a generic model with a traditional vehicle shape, rather than the GTS cab over shape.
3. The team wishes to provide industry with our current status and results and to obtain feedback and guidance from industry as to the program experimental and computational plans. The desired approach is through an exchange of information during site visits and formal presentations at industry attended conferences.

Overview of the Project, Current Funding, and Other Activities

Jules Routbort of DOE OHVT and Argonne National Laboratory provided an overview of the OHVT budget for fiscal year (FY) 2000 and the projected budget for FY01 for heavy vehicle systems aerodynamic drag reduction. The OHVT call for proposals resulted in an additional \$1.025 million being awarded to three companies for projects involving parasitic energy losses for heavy vehicle systems.

OHVT is also interested in providing funding for research and development related to heavy vehicle safety. Much of the meeting discussion focused on the safety issues of splash and spray. Experiments require moving ground planes and a computations would need to model of how a tire picks up water. It was mentioned that DOT has an ongoing project investigating methods for reducing splash and spray. (See related action items at end of report.)

An overview of the project was presented by Rose McCallen of LLNL. The viewgraphs are enclosed. Budget issues were presented as well as the project calendar of events and plans for submitting proposals for needed funding. Discussion at the end of the last meeting day with just the Aero team members resulted in a prioritized list of tasks and the deliverables expected with FY01 projected funding.

Industrial Perspective

Sunil Jain of International Truck and Engine Corporation provided an overview of the current aerodynamic effort at his company. It was emphasized that industry desires cost effective computational fluid dynamics (CFD) tools that they can use to improve the aerodynamics of their vehicles. The acquisition and use of these tools must be a minimum investment, otherwise experimentation will be more feasible. Industry's ultimate goal is to integrate CFD and wind tunnel testing and bring the computations into the design process. The importance of understanding the flow phenomena, determining optimum vehicle shape to minimize drag, and to be able to correlate the computed and experimental drag coefficient was discussed. Included in the presentation was a case where successful comparisons were made of calculations and experiments for a vehicle design with the use of a commercial CFD tool. It was suggested that the DOE Truck Aero Team provide industry with the CFD tools they are using so that they can investigate their use. It was also suggested that the Team establish collaborative relationships with the commercial software industry providing advanced modeling guidance that will enhance those tools now being used by the tractor manufacturers.

NASA's Plans for 7-ft x 10-ft Wind Tunnel Experiments in FY01 and Plans for 12-ft Wind Tunnel Experiments

Jim Ross of NASA Ames presented options for the 7-ft x 10-ft wind tunnel and 12-ft pressure wind tunnel tests which include experiments with a modified GTS model or use of a traditional vehicle (rather than a cab over shape like the GTS model) with more realistic features. NASA's plans also include provisions for USC to test their GTS model in the 7-ft x 10-ft wind tunnel for evaluation at lower blockage and higher Reynolds number flow. The purpose of all these experiments are for validation of the computational fluid dynamics (CFD) models and for further insight into truck flow phenomena. Details of the NASA test plans are provided in the attached viewgraphs. Several action items related to needed input from the Team on testing requirements are outlined in the action item list at the end of this report.

The public release of the NASA test data was also discussed. It is expected that by the end of October a NASA technical memo data report will be publicly released. The Team discussed the need for a special SAE conference session to release our computational and experimental analysis results.

USC's Wind Tunnel Tests and a Look at an Aero Device

Fred Browand of USC provided a detailed presentation of their gap flow and boattail analysis and a plan for future experiments and analysis of results. Also presented were plans for the development and testing of an oscillation device to control the trailer wake flow. The device alters the turbulent structure of the wake resulting in a drag reduction. Details of the analysis results and test plans are provided in the attached viewgraphs.

RANS and DES Computations at SNL

An overview of the Reynolds-averaged Navier Stokes (RANS) computation being performed by SNL was presented by Kambiz Salari. Current efforts involve the modeling of the NASA experiments in the 7-ft x 10-ft wind tunnel. The RANS calculation presented compare well with experiment except for the calculated pressure at the edges of the trailer base. It was recognized that accurate computation of the pressure gradient at the trailing edges of the trailer are important in correctly determining the vehicle drag. The possible need for edge effect corrections by averaging the pressures for the perpendicular element segments at the edge were discussed.

Some preliminary detached-eddy simulation (DES) results for flow around a circular cylinder were also presented. DES is a new turbulence modeling approach where RANS is used in wall regions and LES is used away from walls for reduced grid resolution requirements near walls. Details of the computations and analysis are provided in the attached viewgraphs.

Large-Eddy Simulations using the Finite Element Method at LLNL

The large-eddy simulation (LES) approach being used by LLNL was presented by Dan Flowers for both their compressible and incompressible flow models. The approach and development challenges were presented along with a progress update. Implementation of the incompressible model is complete and some validation remains. See attached viewgraphs for details on the models.

Jerry Owens of LLNL presented an overview of his research in the analysis of time dependent results with movies. Jerry's so called 'movie in the morning' approach for handling large and long running batch jobs, the resulting enormous computational data files, and quick production of movies of this data with overnight turn around was presented. The viewgraphs for this presentation are attached.

Greg Laskowski a student employee at LLNL from Stanford University presented his research and development in DES for incompressible flow modeling using the finite element method. Details are provided in the attached viewgraphs.

Simulations using Vortex Methods: A Gridless Technique

The Caltech group continues to improve their fast, parallelized, adaptive vortex method. Current activities at Caltech include: incorporating bodies with arbitrary complexity, obtaining higher Reynolds numbers computations, and developing and analyzing subgrid models for large-eddy simulation. Mark Brady of Caltech provided an update of their progress in the development and use of the vortex method approach for a two-body tractor-trailer geometry and Tony Leonard provided an overview of his investigation of wall

turbulence models. Simulation Reynolds numbers are still quite low but plans are to move into the higher Reynolds number regime with the addition of subgrid scale models. Details and results of computations with the vortex method code and on the turbulence modeling approach are in the attached viewgraphs.

Demonstration Vehicle

Ross Sheckler of Dynacs Corporation presented his preliminary design for a demonstration vehicle. Ross would like for the Team to choose his design for future testing in the NASA wind tunnels. The pros and cons of the various geometry options were discussed in addition to the possibilities of funding contributions from industry with the appropriate choice of model geometry. Requesting assistance from the Truck Manufacturing Association (TMA) in obtaining industry support was mentioned as a possible approach. Ross's viewgraphs are attached.

Action Items

The follow-on action items with the individuals responsible for the tasks are as follows:

Needed information for NASA experiments (K. Salari)

- Desired gap distance
- Inlet measurements desired (e.g., hot film, rake, PIV)
- Location of unsteady taps
- Are pressure sensitive paint measurements desired
- Determine if investigation of BLA Technology add-on desired.

Provide NASA with time estimate for USC experiments in 7-ft x10-ft wind tunnel (F. Browand)

Establish what model will use in FY01 and FY02 NASA tests (J. Ross)

Provide accuracy of measured C_p (J. Ross)

Complete team budget estimate for FY01 (R. McCallen)

Meeting report with viewgraphs (R. McCallen)

Quarterly report due November 15, 2000 (R. McCallen)

Establish location and schedule next working group meeting (R. McCallen)

Determine relation of splash and spray to vehicle accidents. Possible source of information is UPS. (R. Sheckler)

Gather information on the current R&D effort related splash and spray in DOT. (F. Browand)

Truck Aero Team Meeting

LLNL, Livermore, CA

August 16-17, 2000

Attendee List

Attendee	Organization	e-mail address and phone
Tom Arledge	NASA	tarledge@mail.arc.nasa.gov, 650-604-1604
Mark Brady	Caltech	mbrady@caltech.edu, 626-395-3285
Fred Browand	USC	browand@spock.usc.edu, 213-740-5359
Dan Flowers	LLNL	flowers4@llnl.gov, 925-422-0529
Roxana Greenman	LLNL	greenman2@llnl.gov, 925-424-2501
Sunil Jain	International	Sunil.Jain@nav-international.com, 219-428-3783
Greg Laskowski	LLNL,Stanford	glaskows@hyper.llnl.gov, 925-423-1581
Tony Leonard	Caltech	tony@galcit.caltech.edu, 626-395-4465
Rose McCallen	LLNL	mccallen1@llnl.gov, 925-423-0958
Jerry Owens	LLNL	jlowens@llnl.gov, 925-422-1646
Jim Ross	NASA	jcross@mail.arc.nasa.gov, 650-604-6722
Jules Routbort	ANL/DOE	routbort@anl.gov, 630-252-5065
Walt Rutledge	SNL	whrutle@sandia.gov, 505-844-6548
Kambiz Salari	SNL	ksalari@sandia.gov, 505-844-9836
Ross Sheckler	Dynacs	dynacsny@dynacs.com, 315-626-6800
Ray Smith	LLNL	smith40@llnl.gov, 925-422-7802
Frank Tokarz	LLNL	tokarz1.llnl.gov, 925-423-3459

Agenda

Heavy Vehicle Aerodynamic Drag: Working Group Meeting

Lawrence Livermore National Laboratory, Livermore, CA

August 16 - 17, 2000

Purpose of Meeting

Discussion of project technical and budget issues and proposed strategies to improve budget situation

Review of experimental and computational plans

Presentation of technical details of experimental and computational work in progress

Wednesday, August 16

Introduction to LLNL's Engineering Directorate **Satish Kulkarni**

21st Century Truck Initiative update **Ray Smith**

Meeting introduction and project overview **Rose McCallen**

DOE/OHVT update and budget issues **Jules Routhbort**

Industry perspective **Sunil Jain**

NASA data reduction, analysis, documentation, test plans **Jim Ross, Tom Arledge**

Thursday, August 17

USC test results, plans, and a look at aero devices **Fred Browand**

SNL RANS computations, analysis and DES development **Kambiz Salari, Walt Rutledge**

LLNL LES compressible, incompressible computations and analysis

Dan Flowers, Jerry Owens, Greg Laskowski, Rose McCallen

Caltech vortex method development and computations **Mark Brady, Tony Leonard**

Demonstration project: description and status **Ross Sheckler**

Discussion:

Plans for release of data (e.g., conference session)

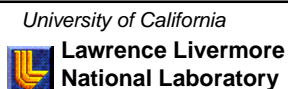
Plans for improved collaboration with tractor, trailer, fleet, and software industry

Discuss needed task expansions (e.g., design) and research for project success
Engineering Foundation Conference
Miscellaneous: David Taylor water tunnel, Clarkson University activities

Aerodynamic Drag Reduction for Class 2, 6, 7 & 8 Trucks

DOE's Aerodynamic Design of Heavy Vehicles Project Team

<http://energy.llnl.gov/aerodrag>



Reducing aerodynamic drag has a higher potential leverage than any other technology improvement.

20 Year Projection

Technology	Fuel Reduction
Improve engine efficiency by 8%	8%
Weight Reduction of 15%	< 10%
Reduce aerodynamic drag by 25%	10 - 15%



Kenworth cab-over-engine (1990)

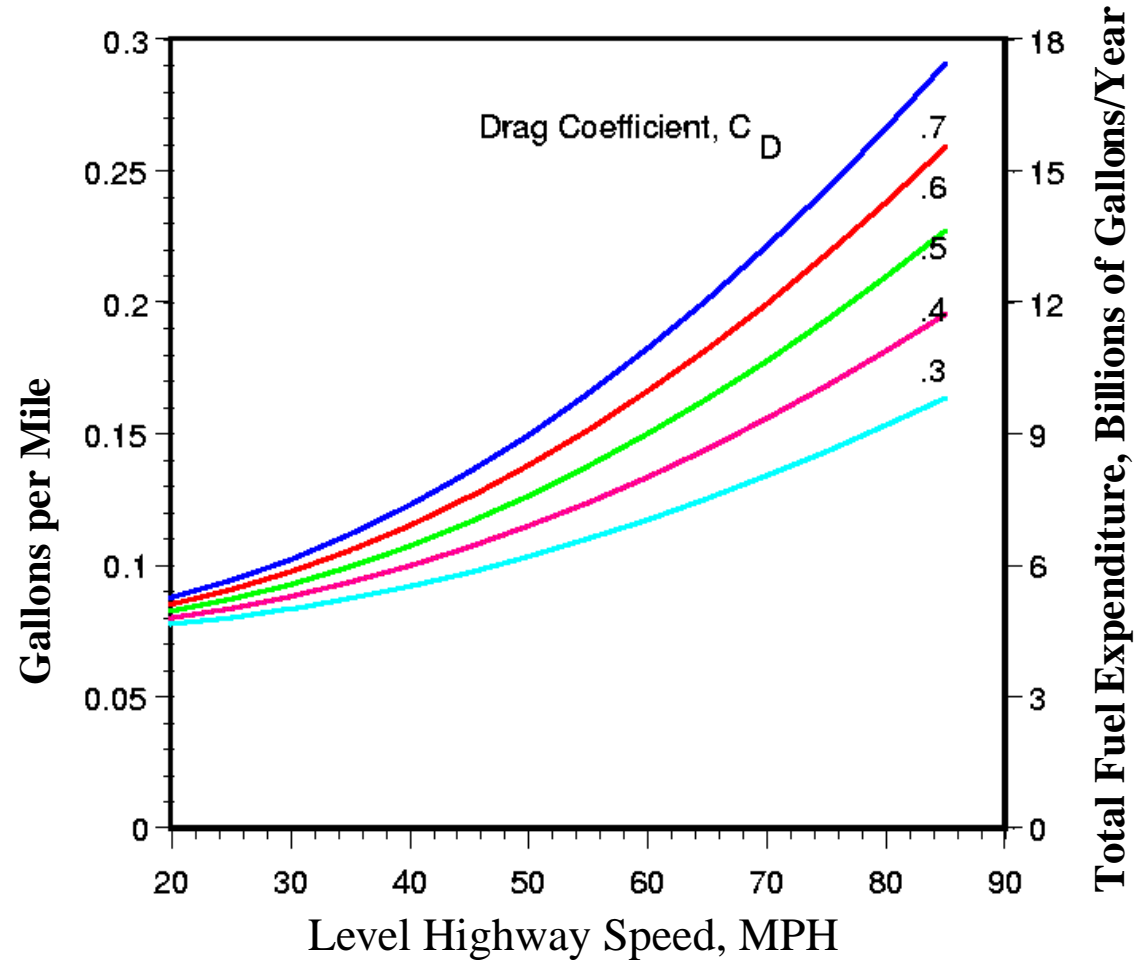


Kenworth T2000 conventional model (1999)

Impact on military versus commercial fuel consumption is dependent on vehicle duty cycles.

Class 8 tractor-trailer
 $C_D=0.6$
speed > 60 mph

All vehicles will benefit from aerodynamic drag reductions – the higher the speed the higher the duration, the most benefit.



The truck industry relies on wind tunnel and field experiments for aerodynamic design and analysis.

Wind Tunnel Testing

Costly detailed models

Expensive tunnel use

Trial-error approach to determine drag effects



Cabover Engine

Field Testing

Performed by both manufacturer and fleet operators



Conventional

Issues

A tractor is paired with several different trailers

Almost no aero design interaction between tractor and trailer manufacturers

The effects of design changes on drag are not well understood and computational guidance is needed

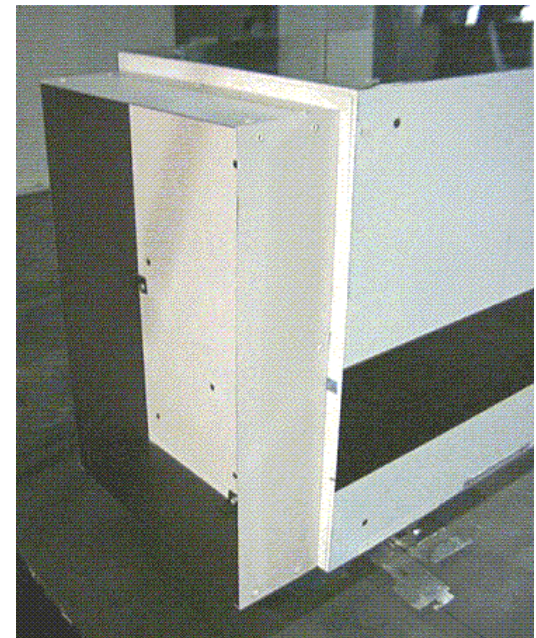
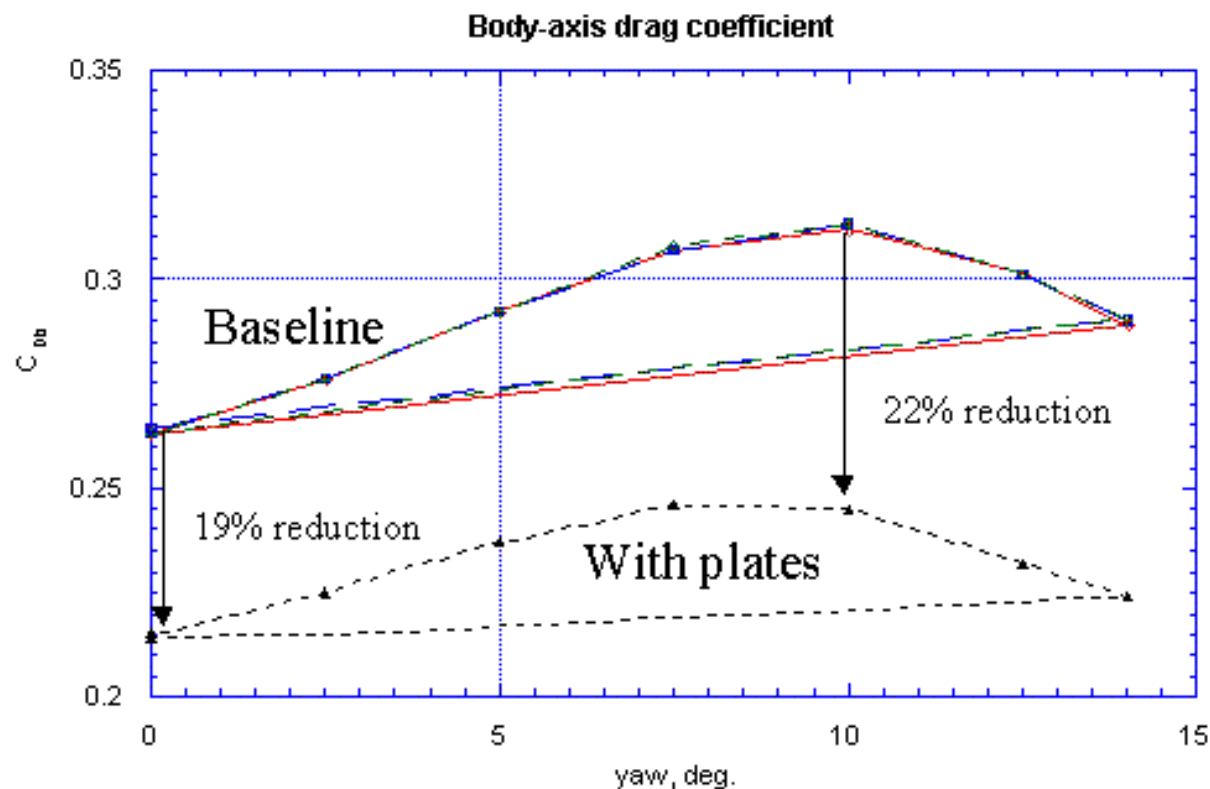
It is possible to realize a 10 to 15% savings in fuel consumption by reducing the drag coefficient by one-quarter.

Example

Boattail plates reduce base drag (wake reduction)

> 20% reduction in wind tunnel on simple model

~ 10% reduction on real truck



Objectives

Develop aerodynamically optimized vehicles which will reduce drag coefficients by

2 - 5% Near Term

4 - 10% Mid Term

5 - 25% Far Term

which will reduce the vehicle fuel consumption by

0.8 - 3% Near Term

1.6 - 6% Mid Term

2 - 15% Far Term

It is possible to realize a 10 to 15% savings in fuel consumption by reducing the drag coefficient by one-quarter.

Percent Improvement in Drag Coefficient

Vehicle Class/Type	Near Term		Mid Term		Far Term	
	Conservative	Aggressive	Conservative	Aggressive	Conservative	Aggressive
2/Utility Truck	2	5	4	10	10	20
6/Enclosed Delivery Truck	<2	<5	<4	<10	5	15
7/Refuse Hauler	<2	<5	<4	<10	5	15
8/Line Haul Rig	2	5	4	10	10	25
8/Dump Truck	<2	<5	<4	<10	5	15

Improve truck safety by reducing the effects of splash and spray and lateral wind loads.

Efficient aerodynamic design leads to less spray



Car disappears behind water spray



1993 Annual Review of Fluid Mechanics
Photos Courtesy of Mercedes-Benz

The large lateral surface area of trucks results in considerable aerodynamic forces for yawed wind loads.



Splash and Spray: Large uptakes of water from the truck tires pose a safety issue.

Goal: Understand mechanisms causing water uptake and spray and determine methods of mitigation

Formulate design rules for the minimization of tire and vehicle spray

Approach: Computational and experimental studies of water spray and splash

Baseline experimental studies - establish fundamental mechanisms

Development of computational tools for water sheet and spray simulation

Laboratory experiments for code validation

Develop guidelines for spray mitigation

Standard Mudguard

Car not visible →



Truck not visible
to driver →

Grooved Mudguard

Reduced water flow between
tire and mudguard

Truck and car can be
seen clearly



Ref. SAE paper 950631

Thermal management: Allow for flexibility of hood and truck design for minimizing drag.

Goal: Improve internal flows for minimum drag and maximum thermal transport

Radiator positioning and design

Improved hood/truck shapes

Approach: Use of computational tools with experimental verification to model underhood flow

Development of coupled flow/thermal transport computational tools

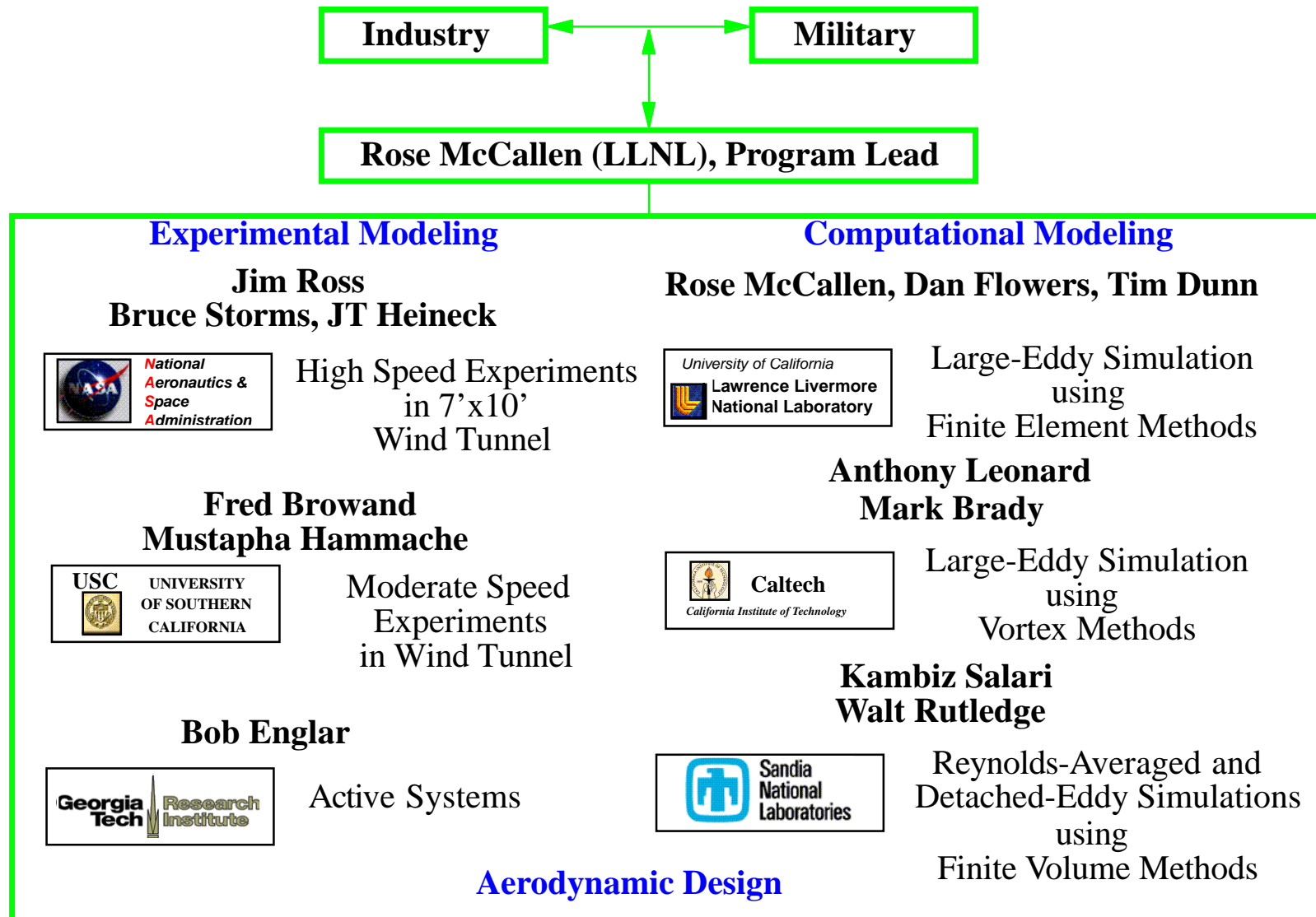
Perform laboratory experiments for code validation

Investigate innovative underhood configurations

DOE Project: Enabling industry to reduce aerodynamic drag on Class 8 trucks.

- Goal:*** Reduce fuel consumption and lower emissions of heavy trucks by reducing aerodynamic drag
- Focus:*** Development and demonstration of a simulation capability
- Computations:*** Computational design capability and
Making wind tunnel testing more effective
- Steady, time averaged RANS modeling (SNL)
 - Unsteady, 3-dimensional LES modeling (LLNL and Caltech)
- Experiments:*** Insight into drag effects
Database for code validation
- High speed wind tunnel testing (NASA)
 - Investigation of tractor-trailer height mismatch and gaps (USC)
 - Past baseline case for code validation at low speeds (SNL/Texas A&M)

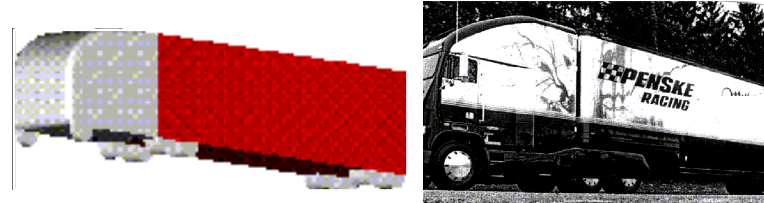
The DOE program has assembled a team of experts and established a working relation with industry.



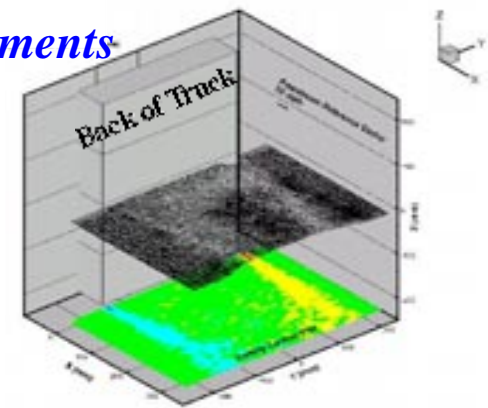
Accomplishments: The team of experts is established and significant progress has been made.

- Established multi-lab, multi-university team
- Working relation with industry
- Multi-year program plan in place
- Experiments completed for baseline case (first time 3D, unsteady velocity field measured in a production wind tunnel)
- Preliminary RANS calculations generated
- Advanced model development and computations in progress

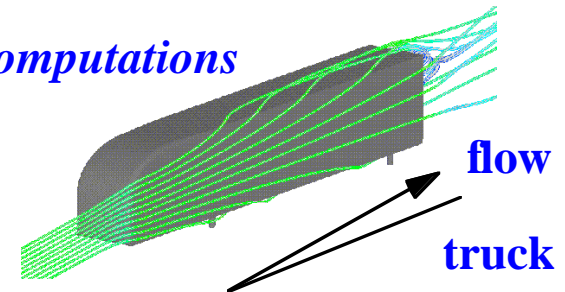
Baseline case



Experiments



Computations



21st Century Truck Initiative

Update to Industry

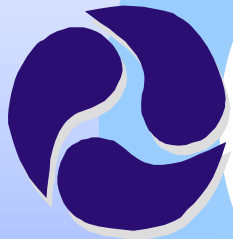
21ST Century Truck Initiative



Department of
Defense



Department of
Energy



Department of
Transportation

21st Century Truck Initiative

Government Agencies and Industry collaborating to improve **fuel efficiency**, reduce **emissions**, increase **safety**, and reduce the **cost of ownership** for the nation's commercial and military trucks.



Industry



Environmental Protection
Agency



Academia

21ST Century Truck Initiative

Fuel Efficiency Goals (mpg)

- **3X - Class 2B & 6**
- **3X - Class 8 Bus**
- **2X - Class 8 Line Haul**

21ST Century Truck Initiative

Technologies to Achieve Aerodynamic Drag Reduction

- Tractor Aerodynamic Drag Improvements
 - Reduce Losses of Base Tractor.
- Tractor-to-Trailer Bridging
 - Reduce Turbulance of Discontinuous Surfaces.
- Tractor / Trailer Underbody Panels
 - Reduce Turbulance of Discontinuous Surfaces.
- Trailer Rear Contoured Panels
 - Streamline Airflow From Trailer.

Aerodynamic Design of Heavy Vehicles

Overview of Project

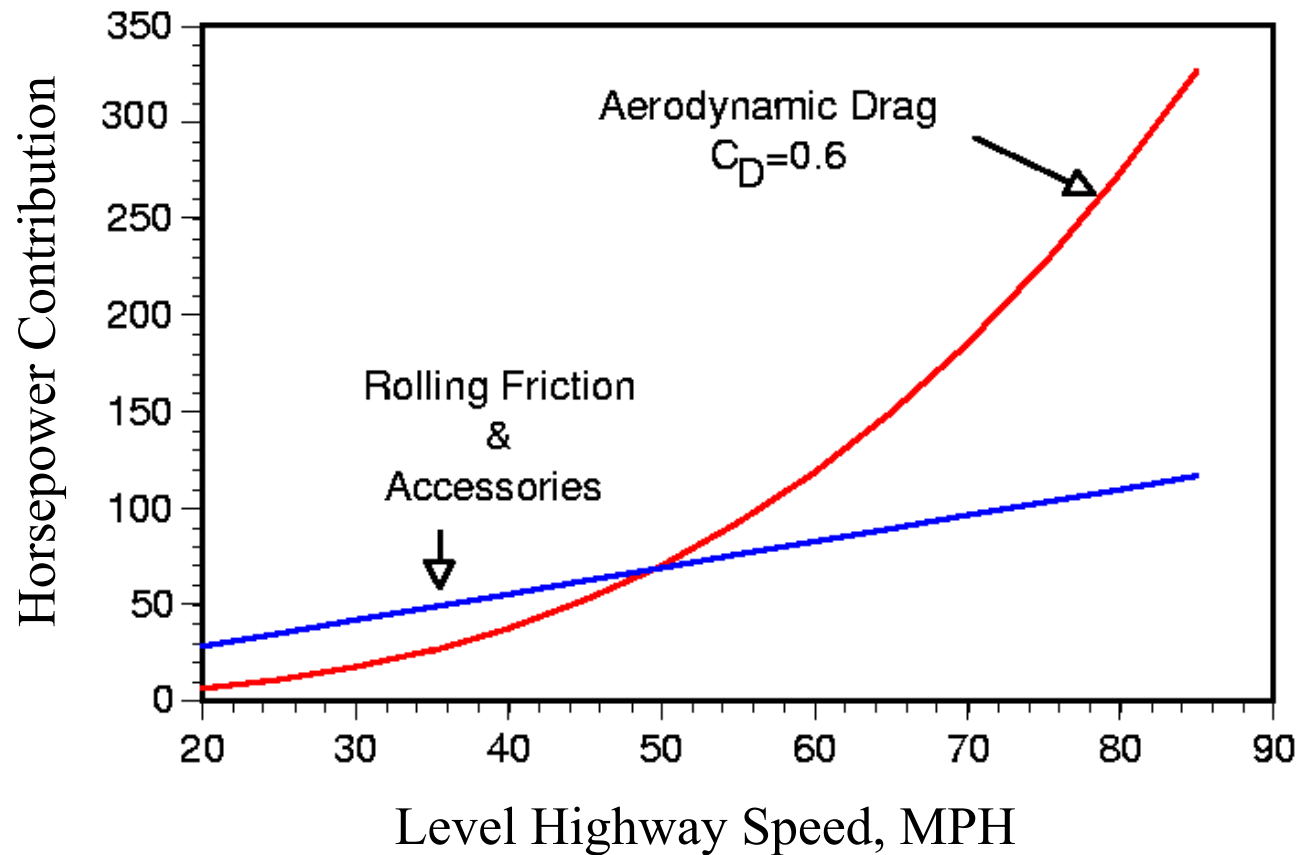
Rose McCallen

Lawrence Livermore National Laboratory, Livermore, CA

August 16, 2000

At 70 mph, 65% of the engine's total energy output is in overcoming aerodynamic drag.

Typical Class 8 tractor-trailer



fuel : C_D reduction is approximately 1:2

Reducing aerodynamic drag has a higher potential leverage than any other technology improvement.

20 Year Projection

Technology	Fuel Reduction
Improve engine efficiency by 8%	8%
Weight reduction of 15%	< 10%
Reduce aerodynamic drag by 25%	10 - 15%

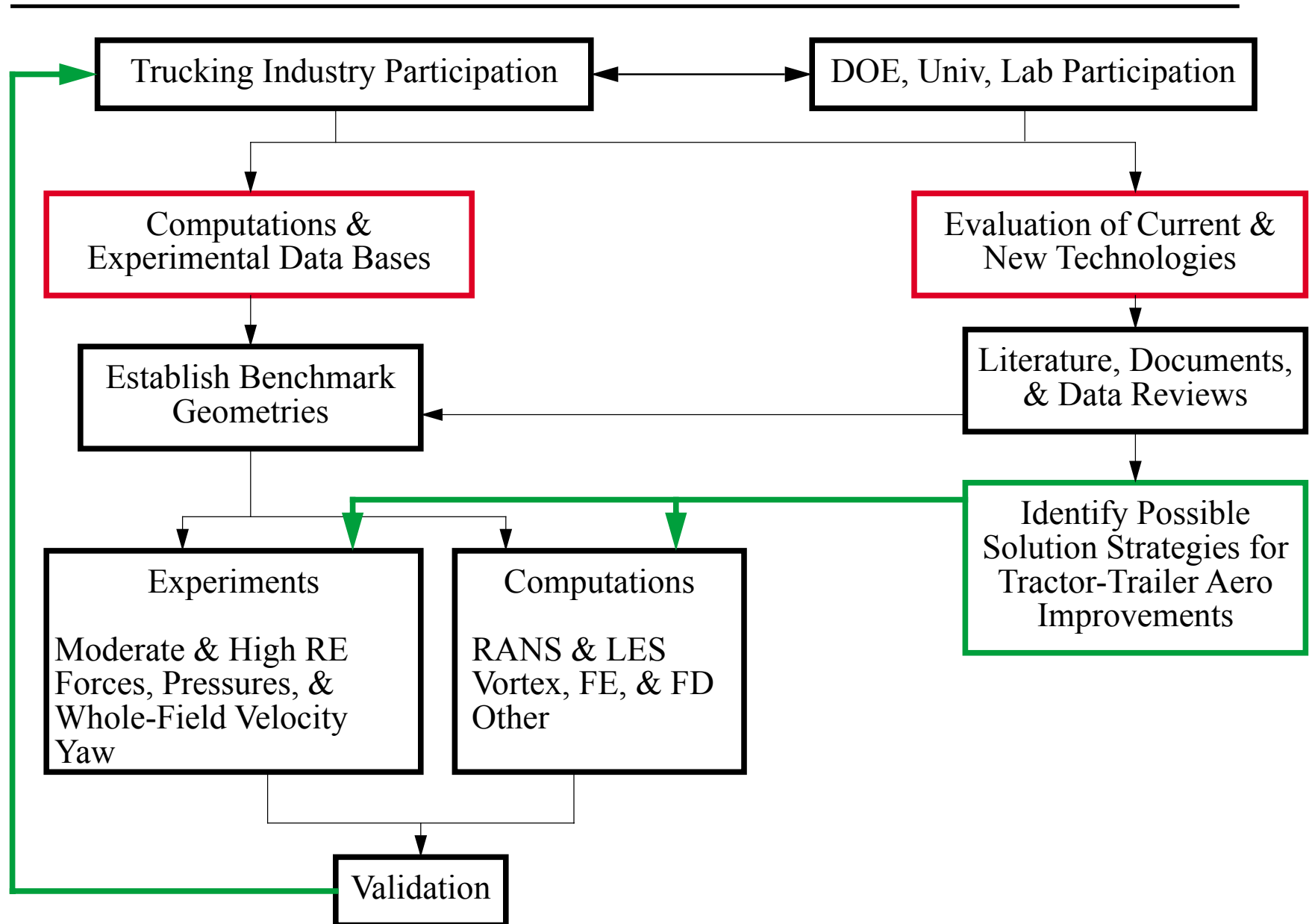


Kenworth cab-over-engine (1990)



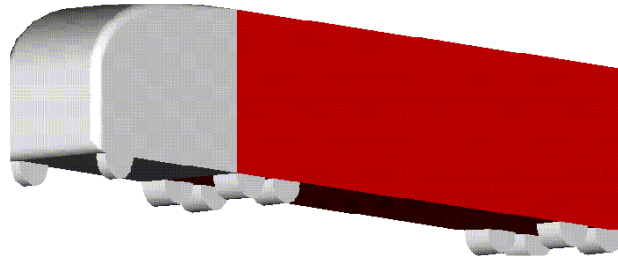
Kenworth T2000 conventional model (1999)

The current project focus is on a validated simulation capability, but the MYPP includes a ‘design’ effort.

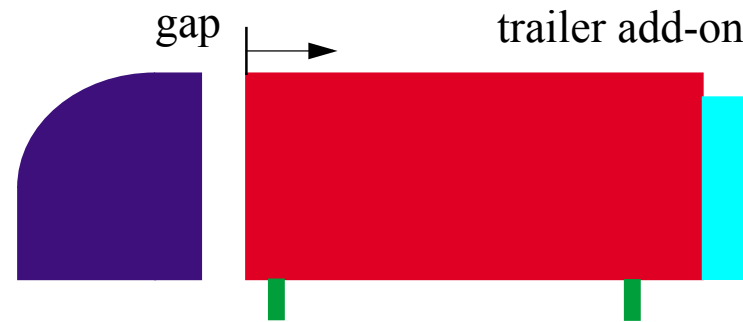


To obtain near-term results on a limited budget, we chose a simple geometry with existing data and modeling.

Ground Transportation System (GTS)



baseline GTS



modified GTS

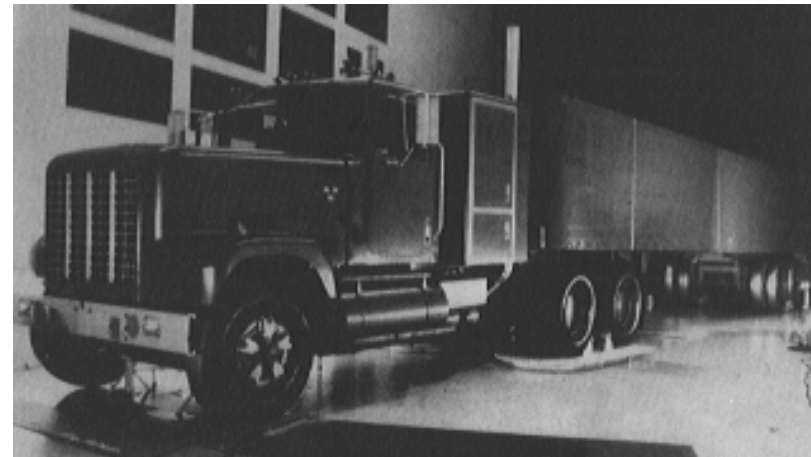
The DOE is interested in improved heavy vehicle thermal management for fuel reduction.

The engine cooling airflow contributes to aerodynamic drag

1970's - 1980's Designs

$$\overline{C}_{Dtotal} = 1.0 - 0.85$$

engine air cooling is 3.8% of \overline{C}_{Dtotal}



Efficient aerodynamic design leads to less splash and spray.



Car disappears behind water spray



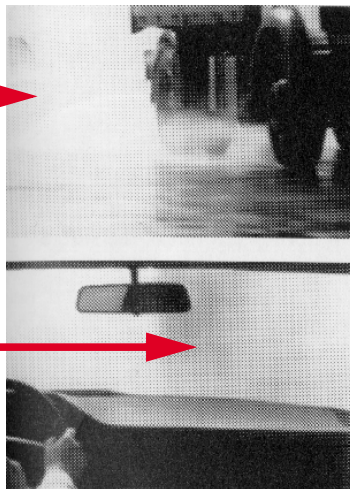
1993 Annual Review of Fluid Mechanics
Photos Courtesy of Mercedes-Benz

Standard Mudguard

car not visible



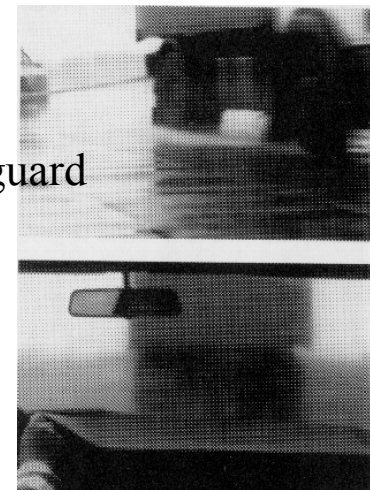
truck not visible
to car driver



Grooved Mudguard

reduced water flow
between tire and mudguard

truck and car can
be seen clearly



Ref. SAE paper 950631

Possible funding sources were investigated, papers/reports were written, and a budget plan constructed.

Progress Report (4/00) ✓

DOE/EE/BES Proposal (4/00) - Assistance from OHVT needed ✓

DOE RFP - Collaborative agreement/cost share with industry not established

Establish scope and industrial partners (WITHOUT TRAVEL)

Pre-application (before 5/1/00), final proposal (before 5/15/00)

DOD RFP - Collaborative agreement/cost share with industry not established

Establish scope and partners - must be complimentary to DOE proposal ✓

White paper (2/29/00) and draft proposal ✓

Final proposal (before 4/28/00)

SAE Meeting, Washington, DC (6/00) ✓

Paper and presentation

Draft FY01 Budget Plan (8/00) ✓

Working Group Meeting (8/00)

Progress Report (7/00) - Meeting report

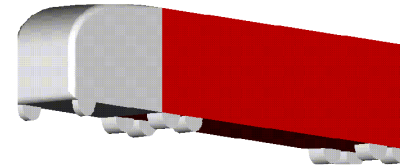
21st Century Truck Meeting (10/28/00)

Truck & Bus Meeting (12/1/00)

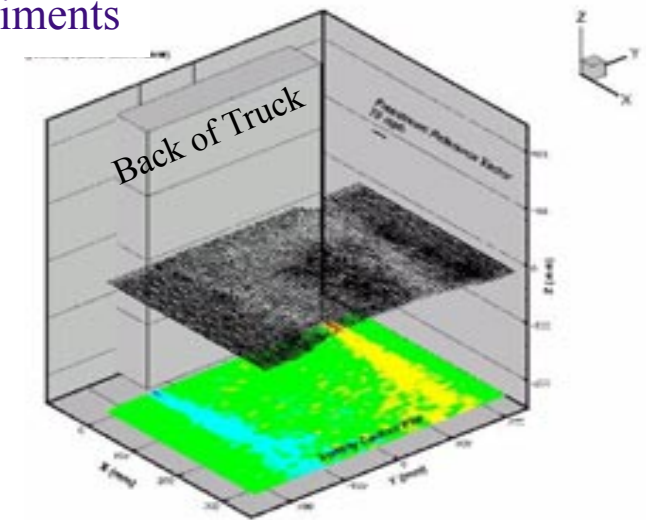
Accomplishments: The team of experts is established and significant progress has been made.

- Established multi-lab, multi-university team
- Working relation with industry
- Multi-year program plan in place
- Experiments completed for baseline case (first time 3D, unsteady velocity field measured in a production wind tunnel)
- Understanding of gap flow phenomena
- Preliminary RANS and LES calculations
- Advanced model development in progress
- Continued data base development
- Preliminary design effort

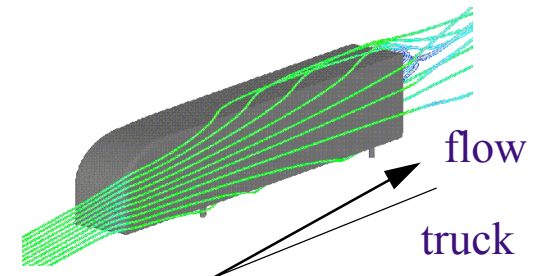
Baseline Case



Experiments

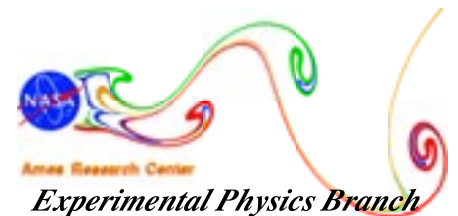


Computations



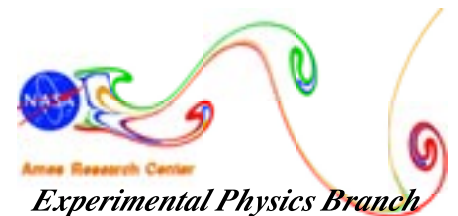
Status and Proposed Activities at Ames

Heavy Vehicle Aerodynamic Drag
Working Group Meeting
8/16/00



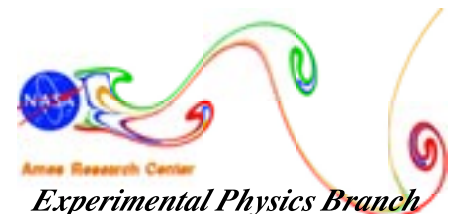
Status

- Report on first 7x10 test complete by end of October
 - Slowed due to conflicts with Ames priorities on Bruce's time
- Second 7x10 test to start in early December
 - Slips in wind-tunnel schedule and intrusion of “high-priority” test
 - Data report by 6/01
- Additional '01 activities TBD



Data Delivered to Date

- Forces and moments for positive yaw only (scale problems)
- Skin friction on top centerline for 0° yaw
 - Dave Driver will process more lines and 10° data in October
- Pressure distributions for numerous cases (referenced to a particular wall pressure)
- PIV data for 4 cases (2 speeds; with and w/o boattails)
- PSP image for 10° yaw (probably not useful for quantitative comparisons)
- Hot-film analysis for 10° yaw showing flow structure
- Two cases of unsteady pressures

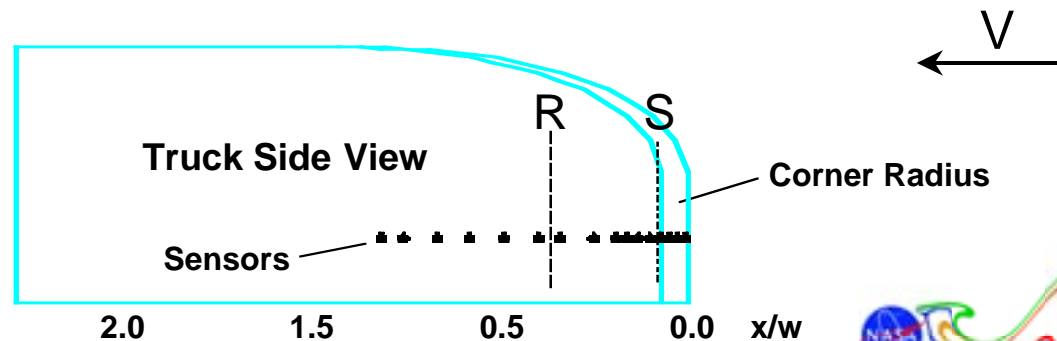
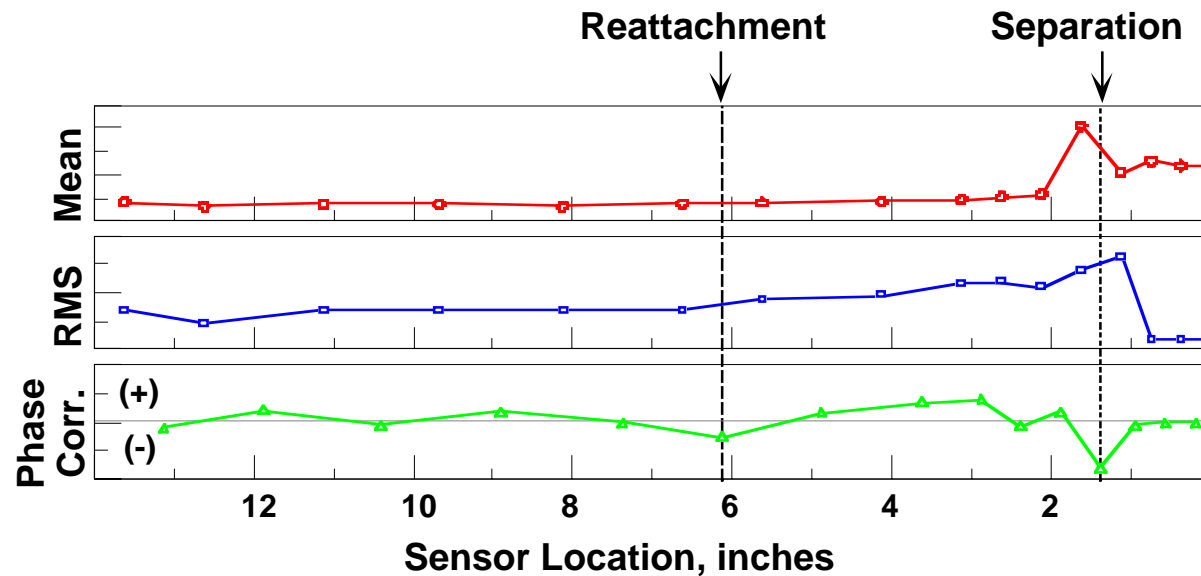


Yet to be Delivered

- Full set of hot-film data, including time histories
- Full set of unsteady pressures
- Full set of C_f
- Data report and archive (on CDROM)

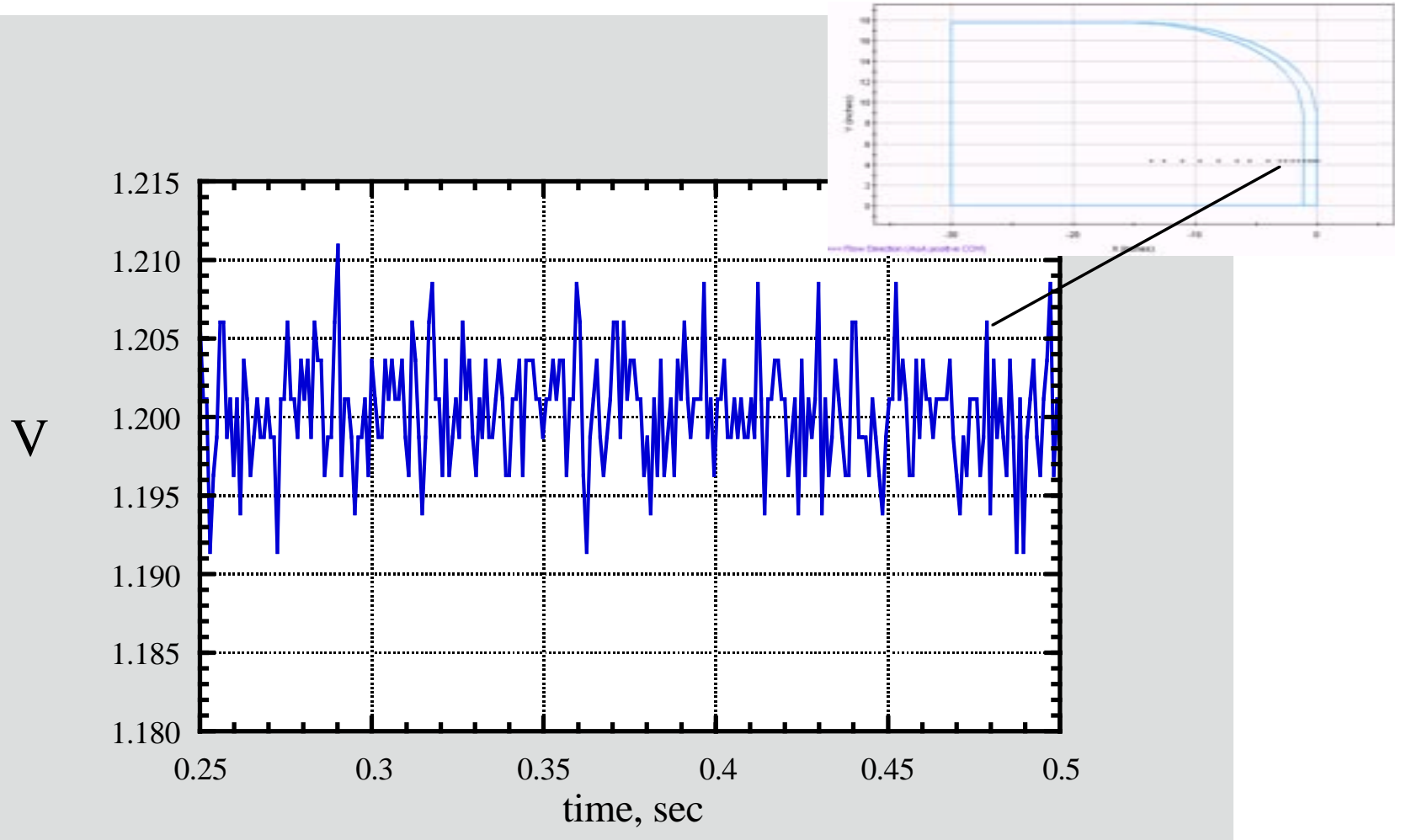
Hot Film Results

No Boattail plates, Yaw = 10 deg, $Re = 2$ million



Hot Film Time History

10° yaw



FY 01 7x10 Experiment (FY '00 funding)

- Modified GTS model
 - Include gap/side extenders for higher-Re CFD validation
 - Additional base-drag reduction device demonstration
- Opportunity to test USC model at lower blockage and higher Re

Mods to GTS Model



Side view

Cut gap in model sized appropriately for long-haul trucks (*what is that gap?*)

Instrument gap and side extenders to measure mean and fluctuating pressure



Top view

Side extenders to model cavity on trucks
- transparent set to allow PIV measurements
- instrumented set for mean and unsteady pressure

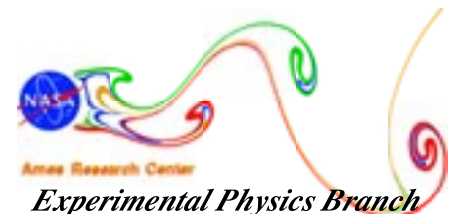
Should we make a lip for the top of the trailer?

Will fabricate a set of plates to allow restoring model to GTS configuration



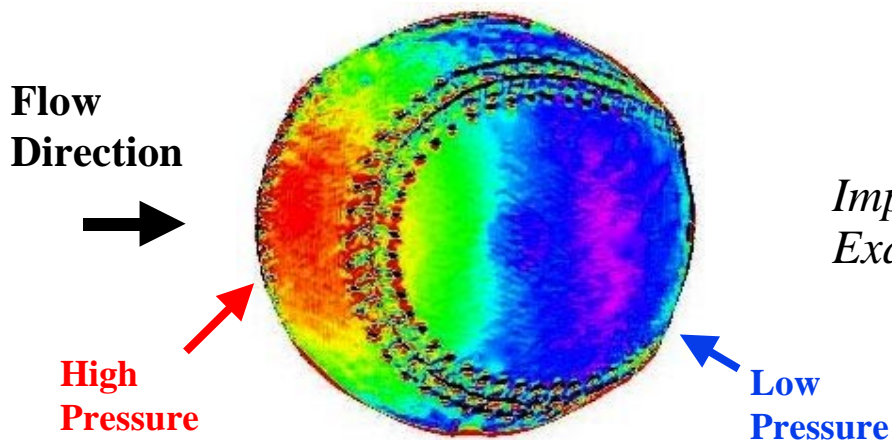
Test Matrix

- Side extender variations for drag (2 lengths plus no extender)
- Select 1 extender length for PIV and other detailed measurements (plus no extender)
 - At least 2 yaw angles
- Repeat drag data for baseline GTS
- Examine drag reduction with and without gap for
 - CDI boat-tail plates
 - BLA drag plate
 - Other drag-reduction devices?
- If noticeable difference in drag reduction with gap, perform PIV measurements in wake to diagnose



Measurements

- Pressure distributions
- Oil film skin friction (limited configurations)
- 3-D PIV in gap (and possibly in wake)
- Mean and unsteady pressures in gap area and rear door
- No PSP unless big demand

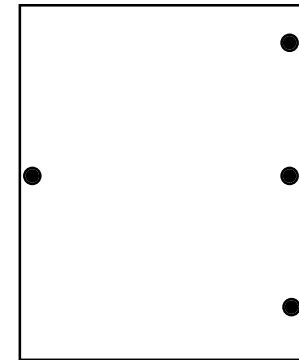


*Improved capability since previous test
Example of 100 ft/sec data*

Unsteady Pressure Locations



Unsteady transducers at rear
of side extenders on inside
surface

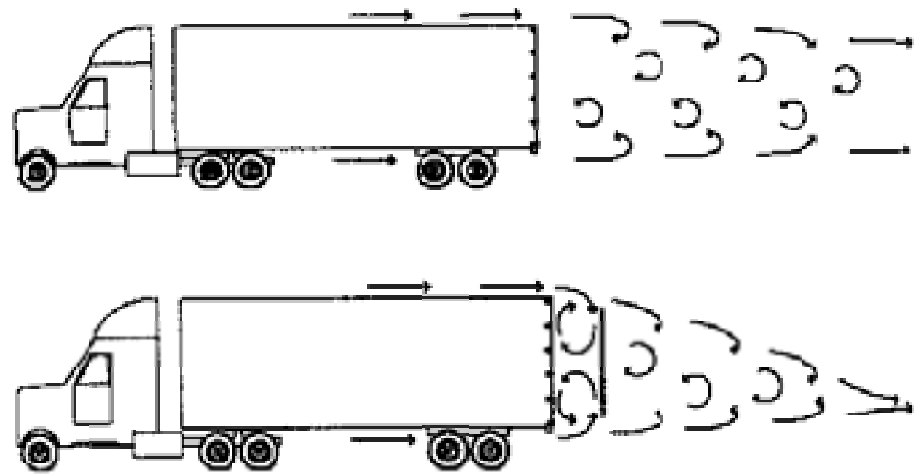
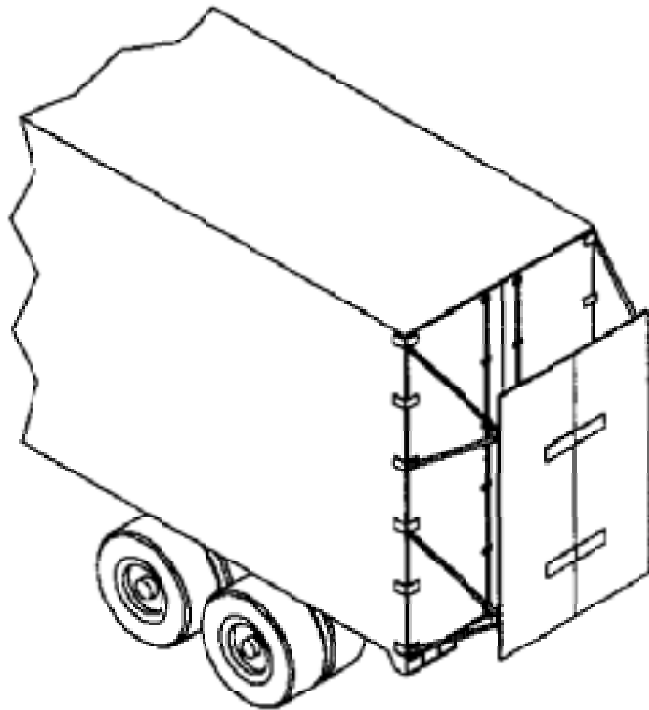


Pattern for unsteady transducers
on back of tractor, front of
trailer, and back of trailer

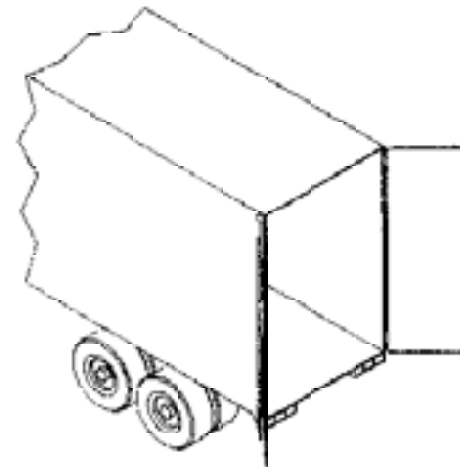
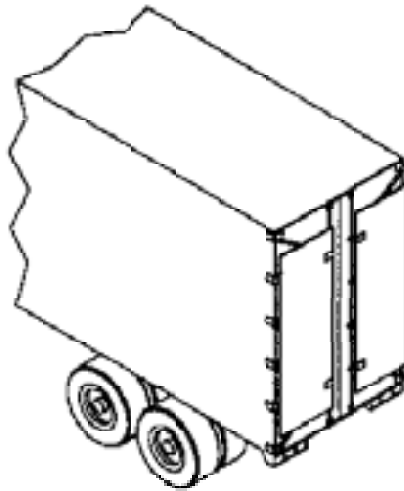
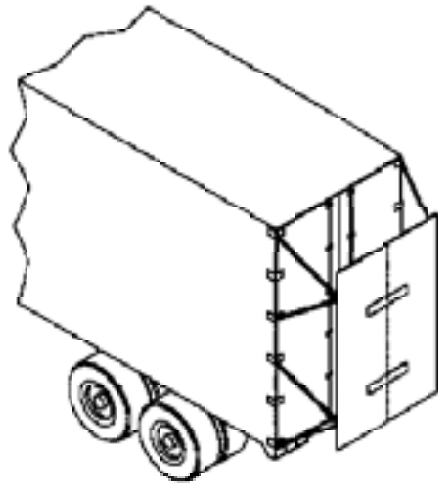
*Will use Endevco 15psia transducers
calibrated using piston phone*

Aeroplate™ Concept

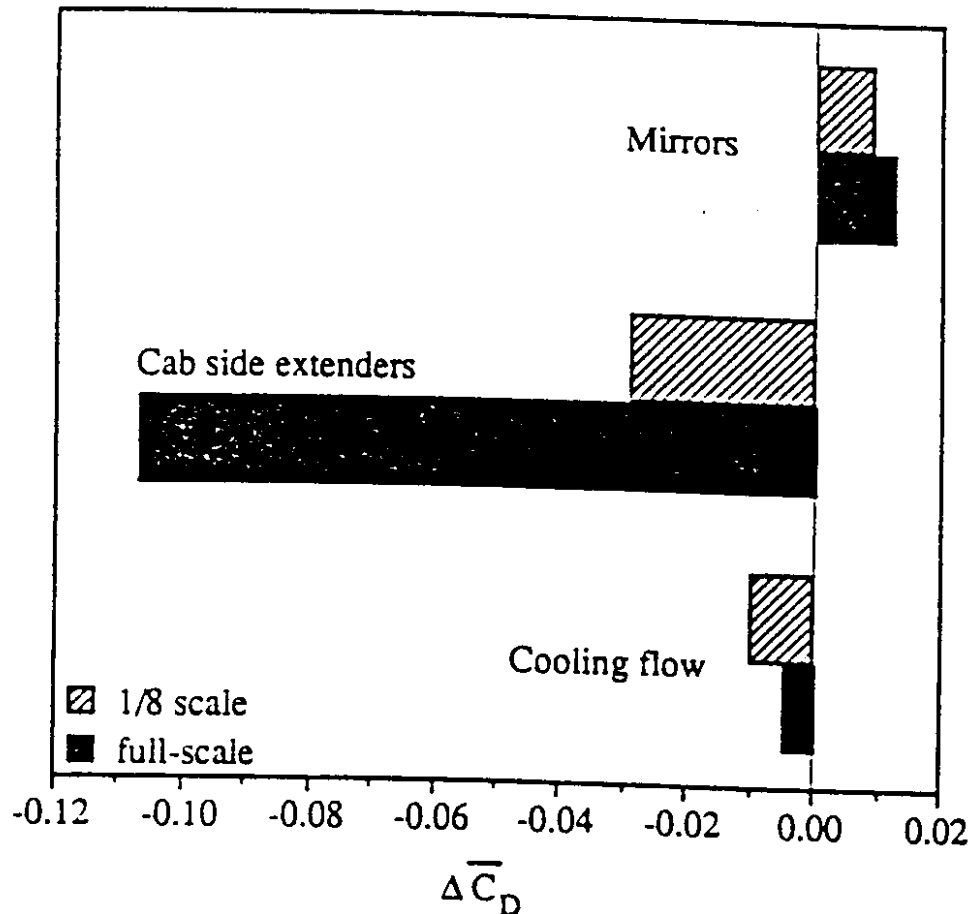
- Developed by B.L.A. Technologies



Aeroplate™ Folding

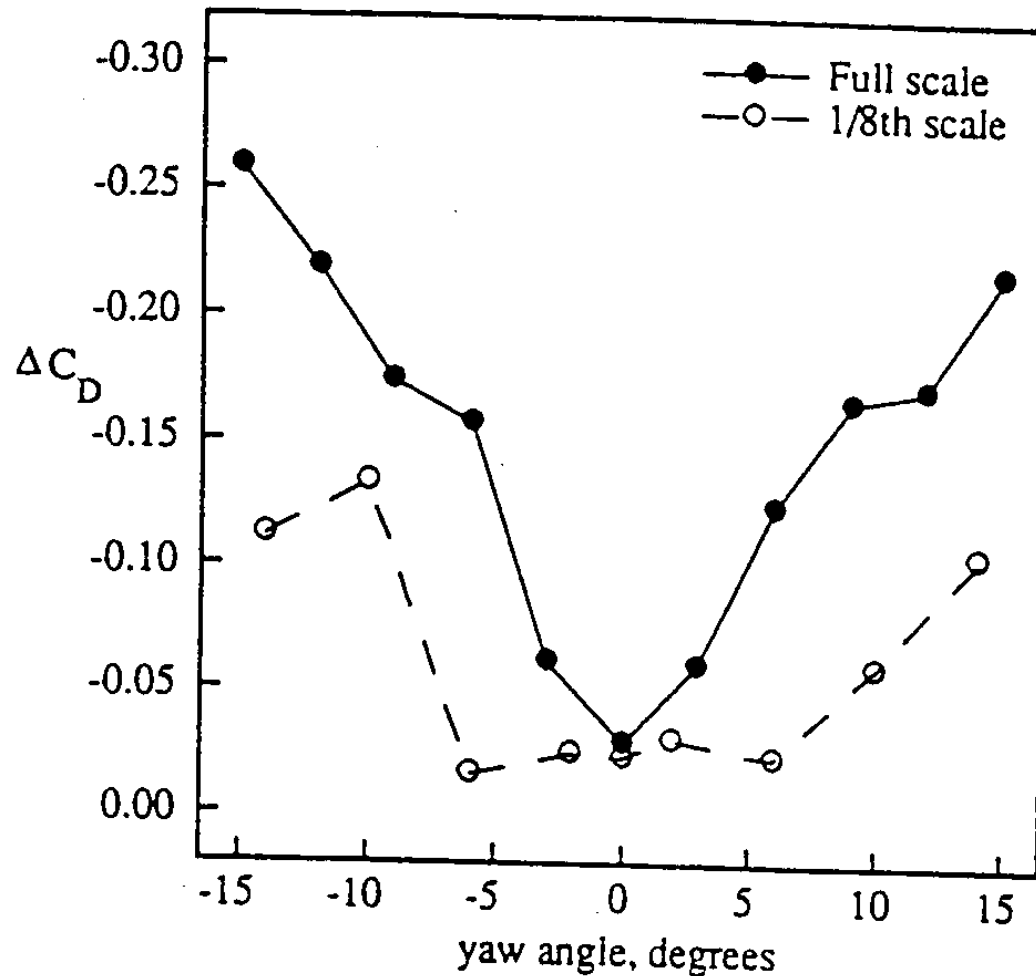


What We Don't Know About Re



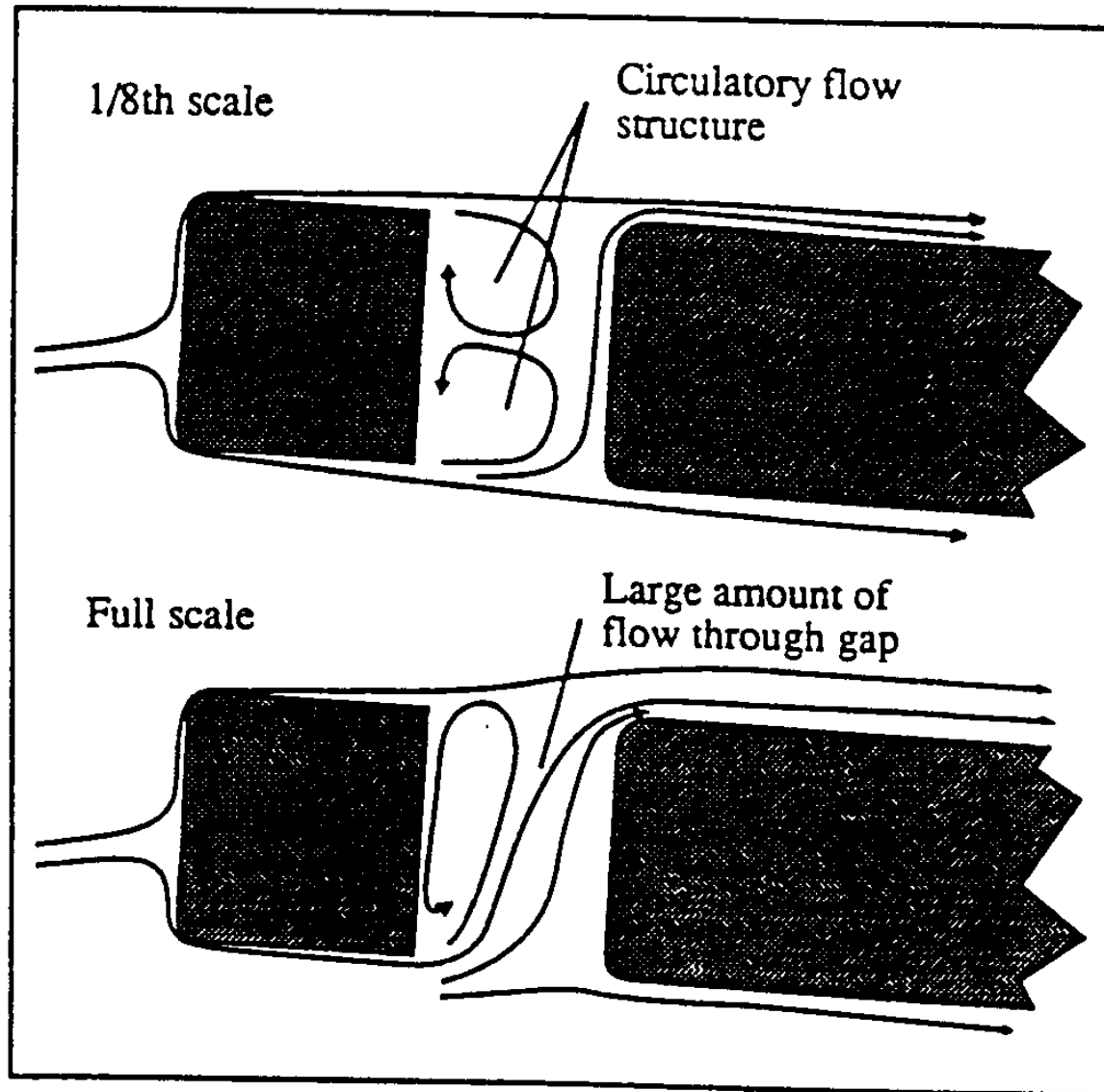
- From full-scale tests with CDI and Navistar
- Cab side extenders had largest mismatch between 1/8 and full scale (wind-averaged drag deltas)
- Other two on plot were smaller and probably affected by geometric details as well as Re change

Drag Change due to Side Extenders versus Yaw



- Drag differences are largest for yawed conditions
- Must be related to gap flow

Suspected Flow State Variation with Re



- At low Re the recirculating flow in gap may be more stable than at high Re
- The stable structure inhibits flow through gap
 - Acts like side extenders
- Result is less change in drag due to side extenders when yawed at low Re than at high Re

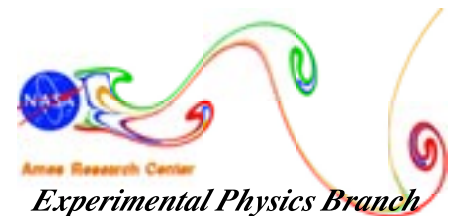
12' Experiment Objectives

- Re effect on wind-averaged drag
 - Range from 0.5 to 5 million (based on width)
- Re effect on drag components - mirrors, bars, *gap/side extenders*, base-drag-reduction devices, etc.
- CFD validation data
 - Skin friction
 - Pressure distributions (PSP works great at high pressure)
 - Unsteady pressures
 - PIV possible

Model for 12' Experiment

Provided by ?

- International routinely tests to ~140 mph (50 psf) (at 60 mph, dynamic pressure is 9 psf)
- Full-scale Re obtained in 12' at 6 atm (80 mph or q of 96 psf)
- Most model parts meet safety factor of 4 - those that don't can be replaced
- Balance arrangement like that used for America's Cup keel or fabricate new single- or two-axis balance (not an expensive proposition)
 - » Max drag < 400#
 - » Max side force < drag
- Yaw range of $\pm 14^\circ$ for wind averaging
- Generic model with CAD definition



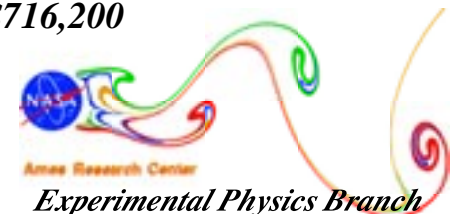
Cost Breakdown for 12' Experiment

What it takes to do the job right

Item	Explanation	Cost basis	Per	Amount	Item cost
Direct Labor	Contracted labor research engineer	\$10,000	FTE month	8.0	\$80,000
Program Support	Head tax for Experimental Physics personnel	\$40,000	FTE year	1.5	\$60,000
Wind Tunnel Time	12' PWT charges	\$160,000	week	3.0	\$480,000
Power Cost	Cost of power to run wind tunnel	\$10,000	week	3.0	\$30,000
Model Mods	Strengthen parts, balance mods	\$15,000	each	1.0	\$15,000
Instrumentation:					
	PIV insertion optics	\$30,000	each	1.0	\$30,000
	Laser installation	\$15,000	each	1.0	\$15,000
	Camera housings	\$10,000	each	2.0	\$20,000
	Second laser for fwd scatter	\$40,000	each	1.0	\$40,000
	Seeder installation	\$25,000	each	1.0	\$25,000
	PSP	\$12,000	each	1.0	\$12,000
				<i>Instr. Total</i>	\$142,000
Center taxes	Directorate, Division, & Branch taxes	3%	of net	\$880,000	\$26,400
"Handling fee"	Tax on reimbursable \$	6%	of net	\$880,000	\$52,800
				Total	\$886,200

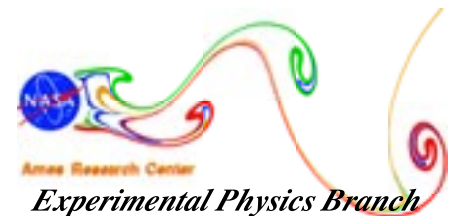
Total for 2 week test \$716,200

Working on a 40% reduction in facility charge
Reduces cost of 3-week test to ~\$700K



Possible CFD Work at Ames

- OVERFLOW computations of geometry with gap & side extenders at 2 yaw angles and 2 Re
 - ~2 month effort
- Donovan Mathias can do work - running ship air wake computations for Navy
- Similar to Navy Ship Airwake computations currently underway

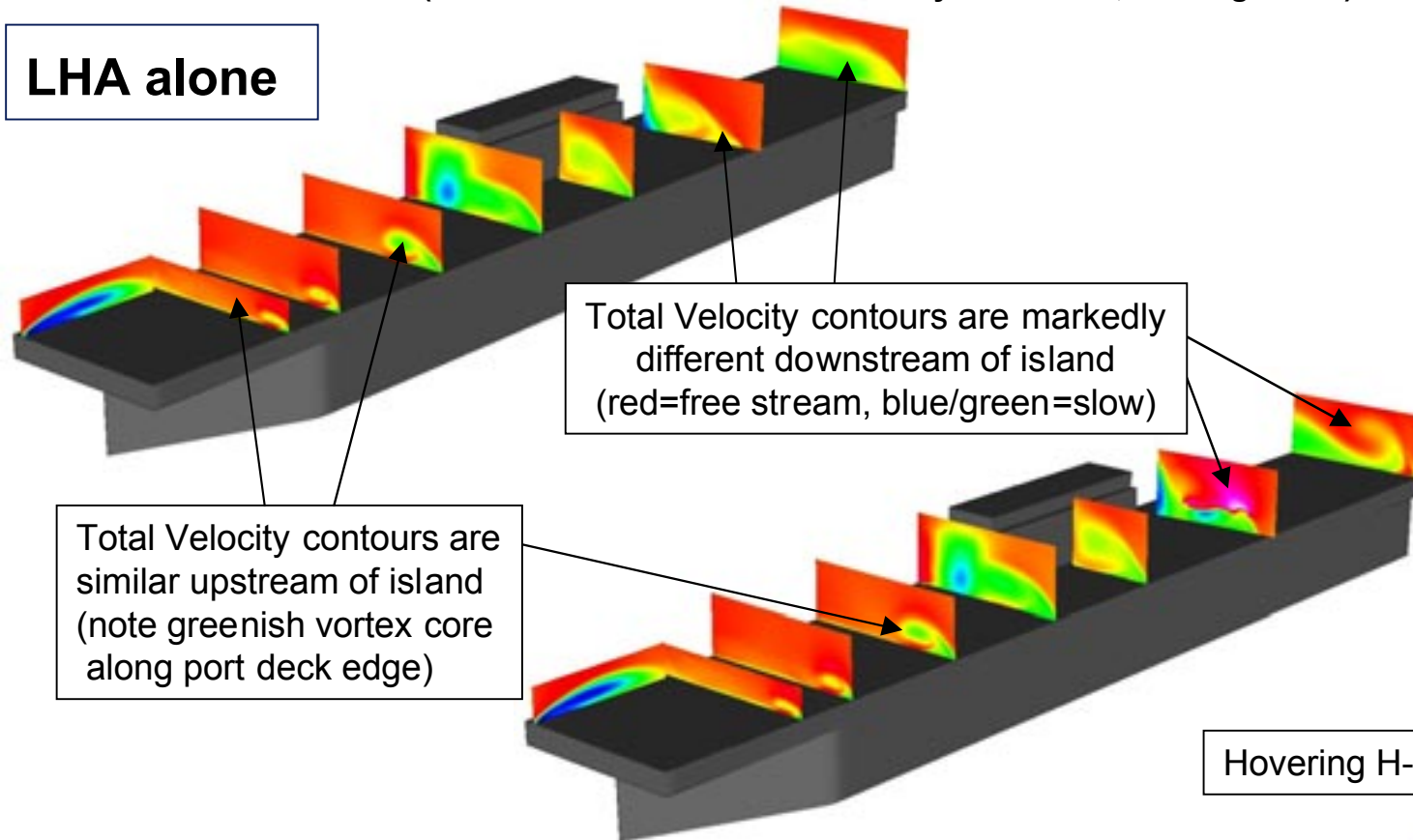




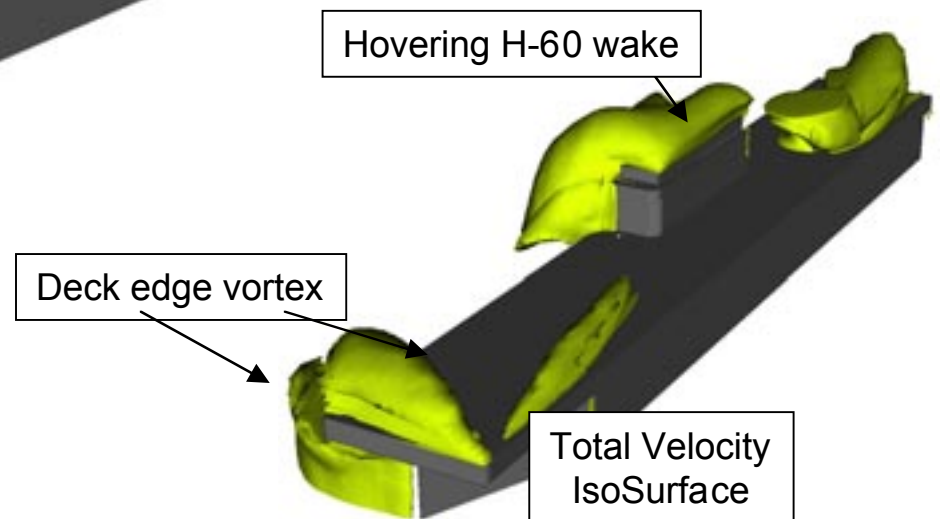
Effect of Hovering H-60 Rotor Thrust on Future LHA¹ Flow Patterns

(Structured RANS CFD Solutions by D. Mathias, 340 deg winds)

LHA alone



LHA + H-60 hovering at spot 7





Drag of Heavy Vehicles

**DOE
Office of Transportation Technology
Heavy Vehicle Systems**

M. Hammache, staff
M. Michaelian, staff
A. Knight, grad student
D. Lazzara, student
P. Kassouf, student
R. Blackwelder, staff
F. Browand, staff
P. Lissaman, staff

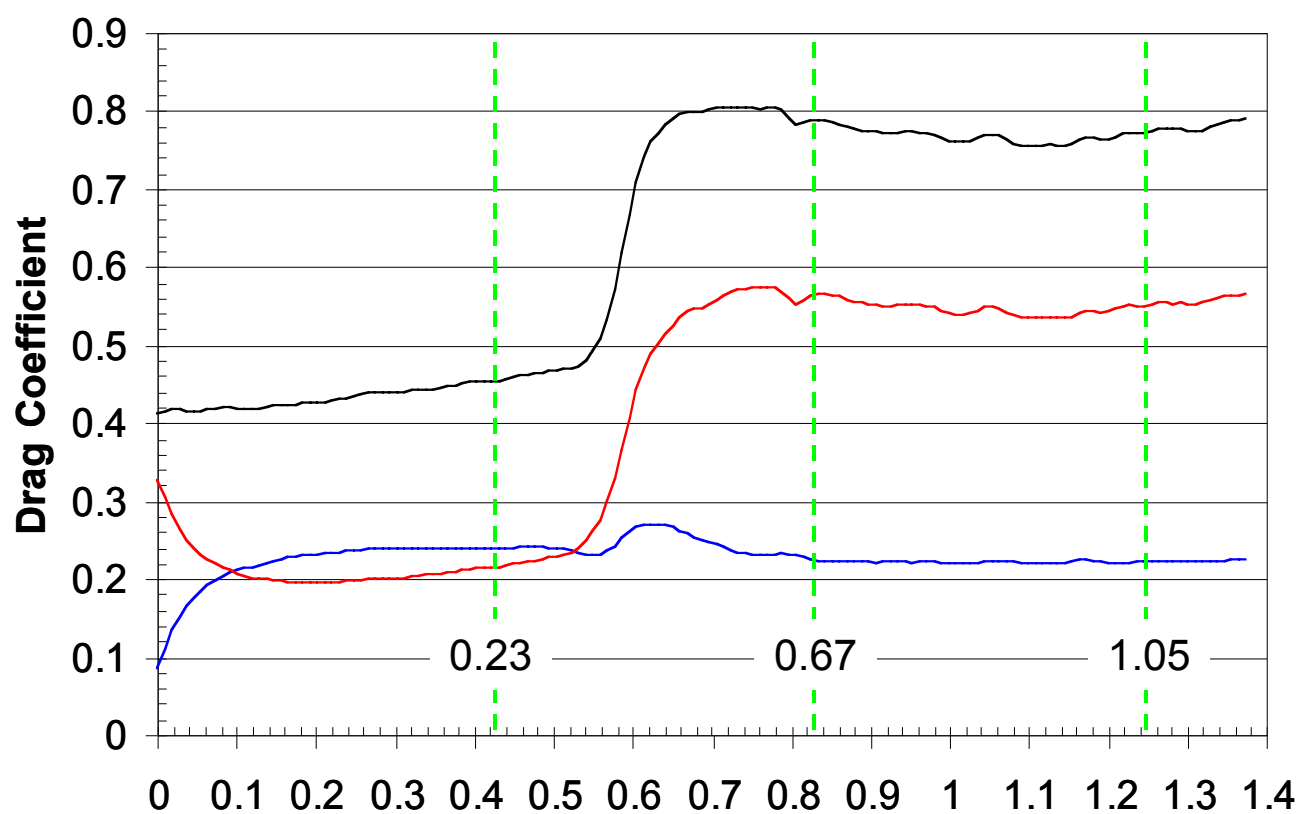
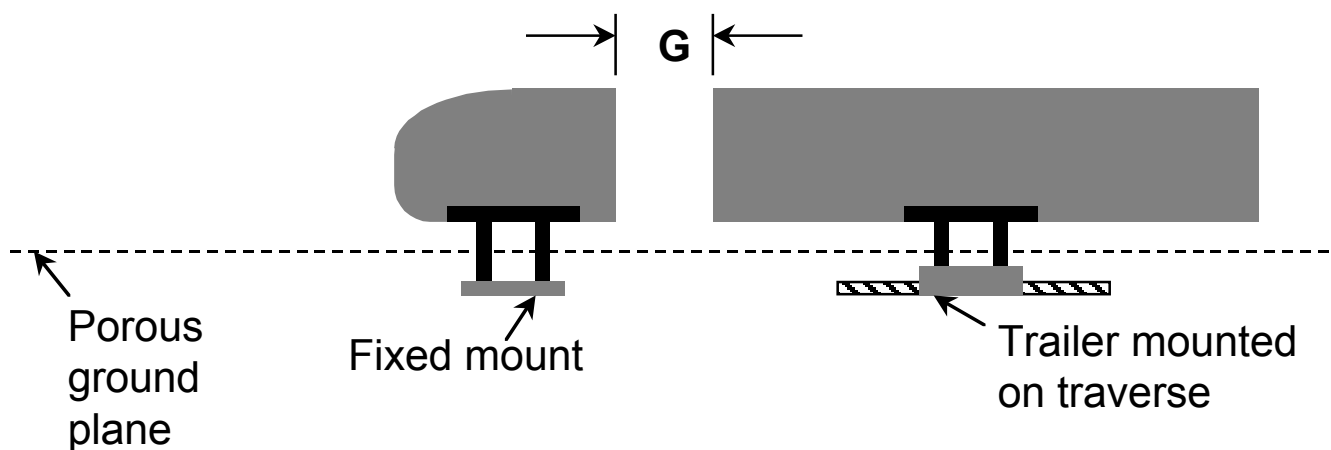
Summary

▲ Truck with gap & yaw

- Qualitative flow visualization
- Force Measurements, $\pm 15^\circ$
- DPIV Results

▲ Flow at rear of trailer w/wo boattail, NASA data base

▲ Future plans

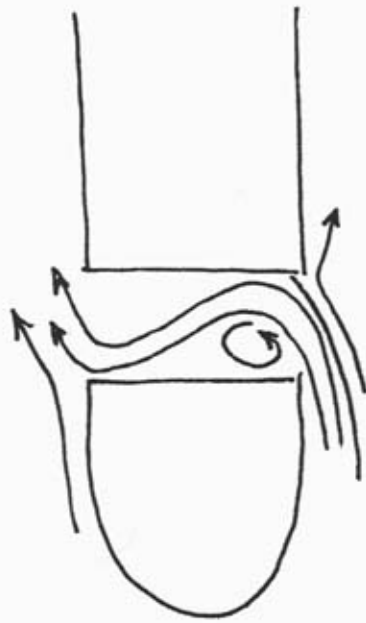


$$\text{Re} = \frac{U \sqrt{A}}{\nu} = 330,000$$

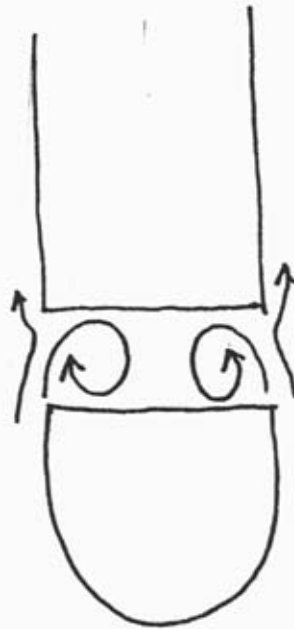
$$\text{Normalized Gap Width} \quad \frac{G}{\sqrt{A}}$$

Conclusions for Gap Flow

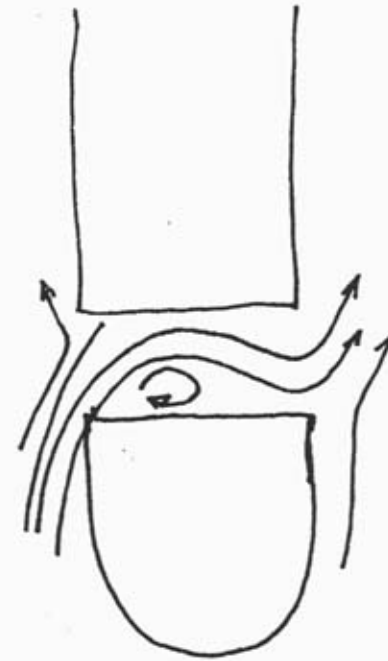
- A critical gap exists at $G/L \approx 0.5$
- For $G/L \leq 0.5$, the gap flow consists of a relatively stable, symmetric toroidal vortex
- A relatively low drag is obtained
- For $G/L \geq 0.5$, the gap cannot support the steady vortex
- The vortex is alternately shed from the gap region, in an unsteady manner
- The relatively smooth flow about the trailer (and tractor) is disrupted, and a large drag results



CRITICAL GAP

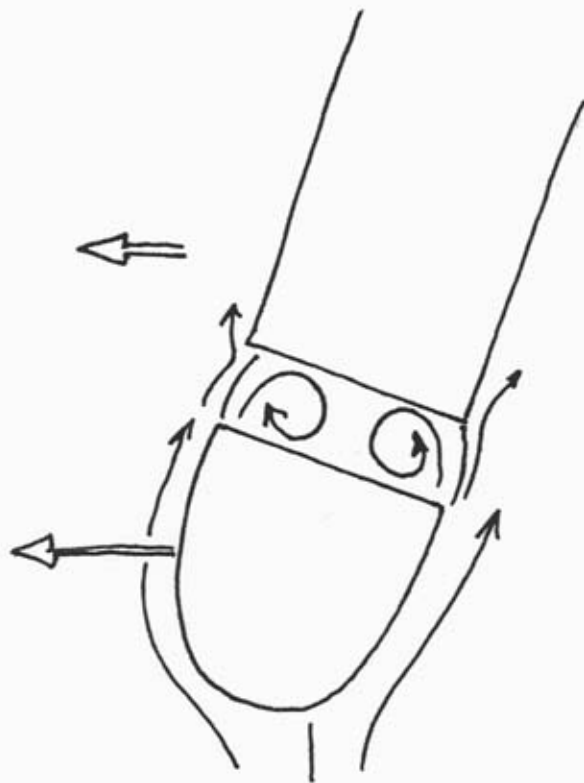


SNAIL GAP

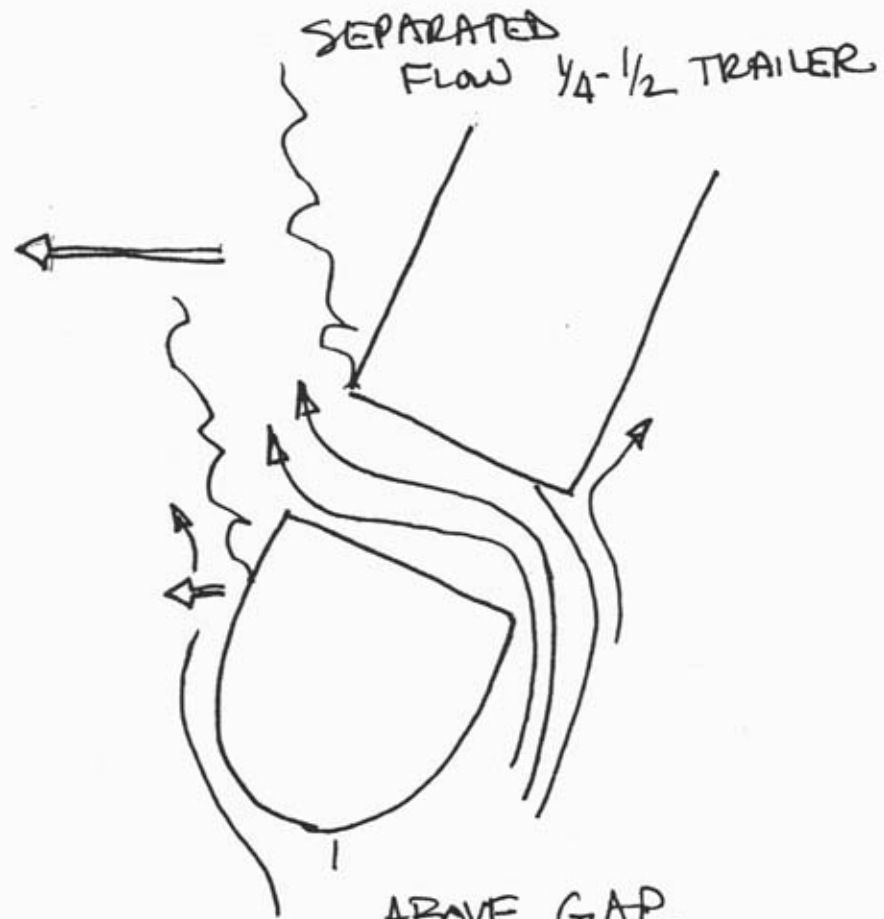


CRITICAL GAP

- ▲ CRITICAL GAP $\cong 0.5\sqrt{A} \equiv GAP_c$
- ▲ THREE "STATES" ARE INTERMITTANTLY PRESENT
- ▲ SAME QUALITATIVE FLOW PICTURES 0-2° YAW
- ▲ AT GAP > GAP_c, TRANSITIONS BECAME LESS VIOLENT

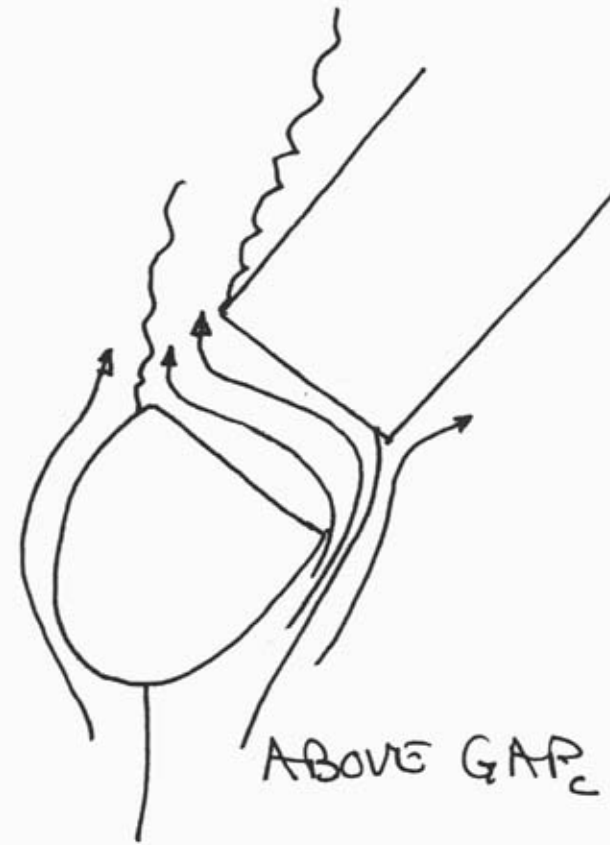
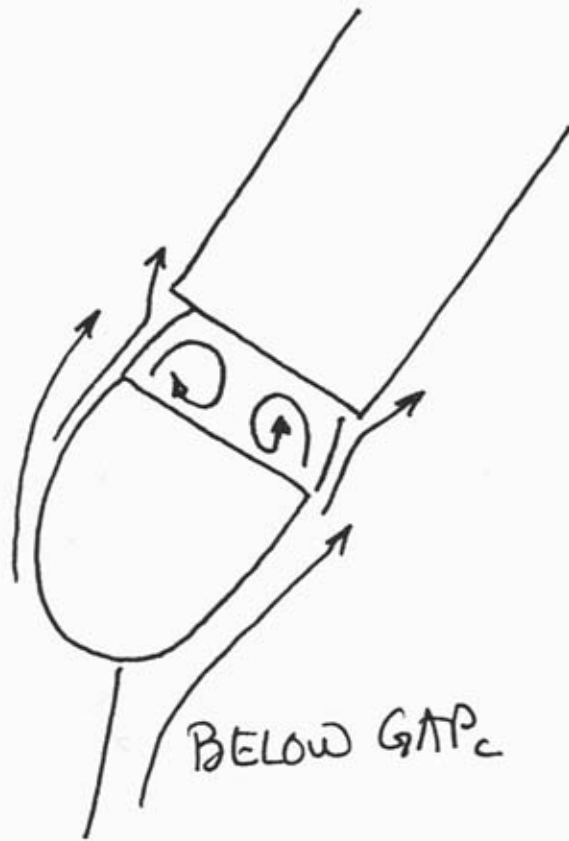


BELOW GAP_c

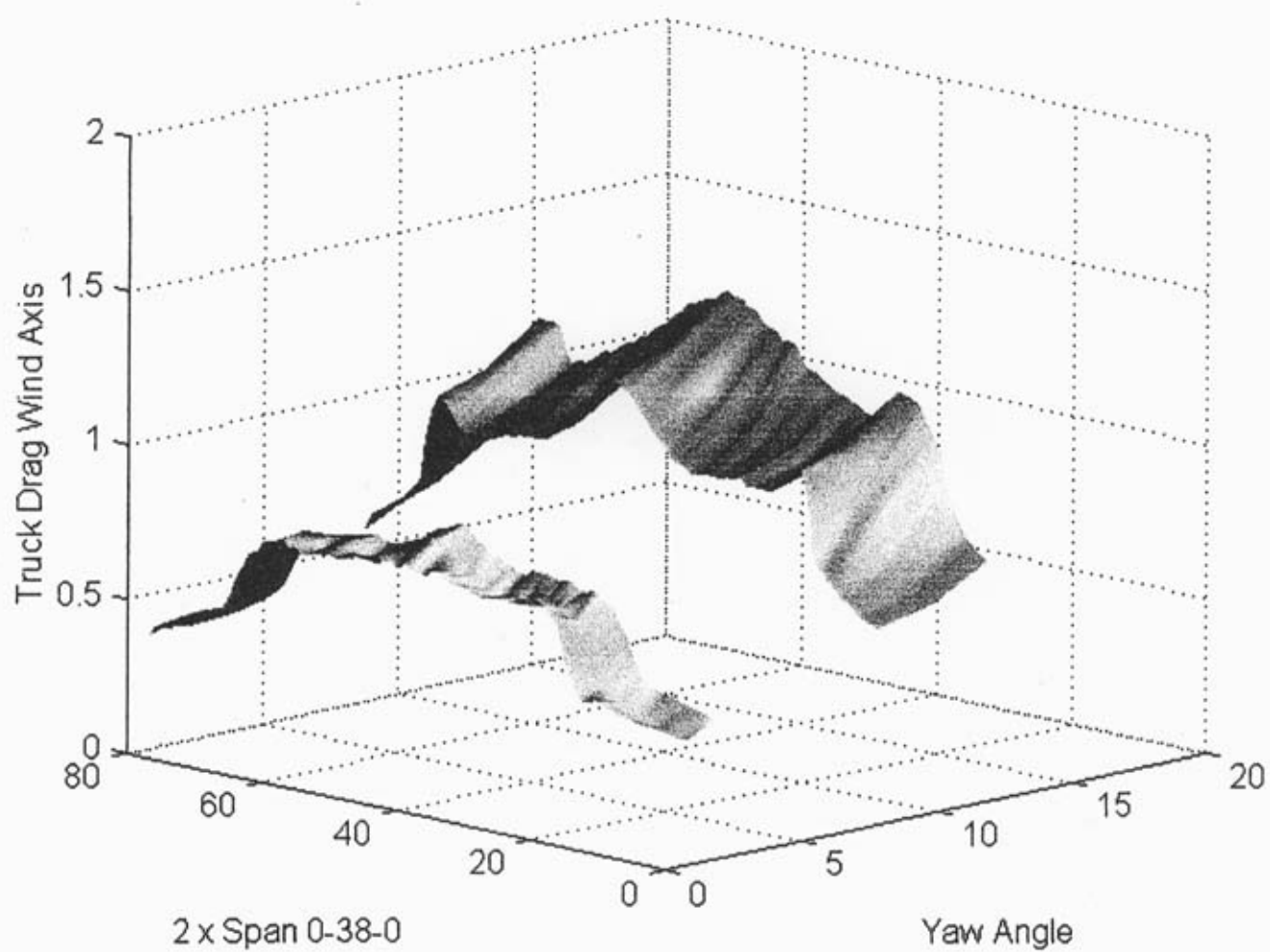


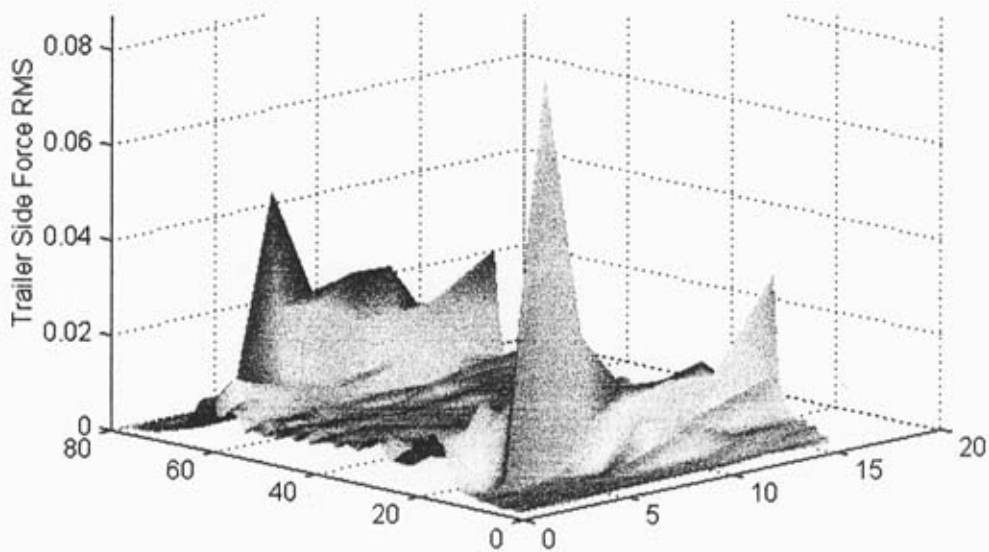
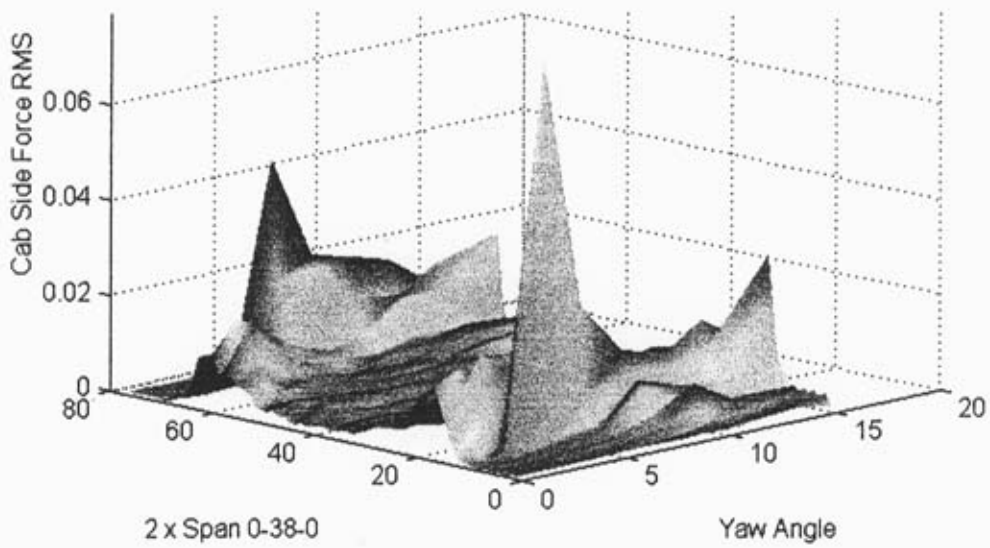
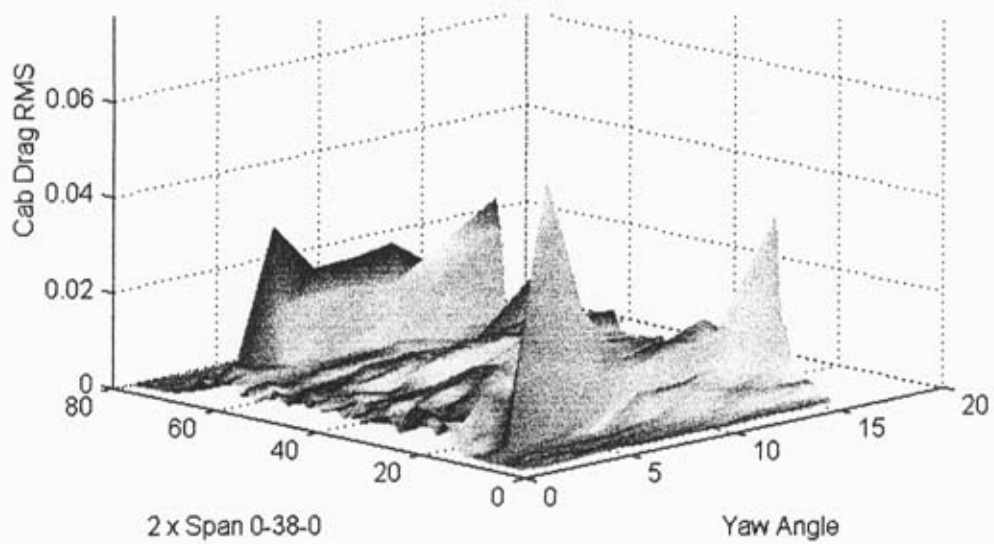
ABOVE GAP_c

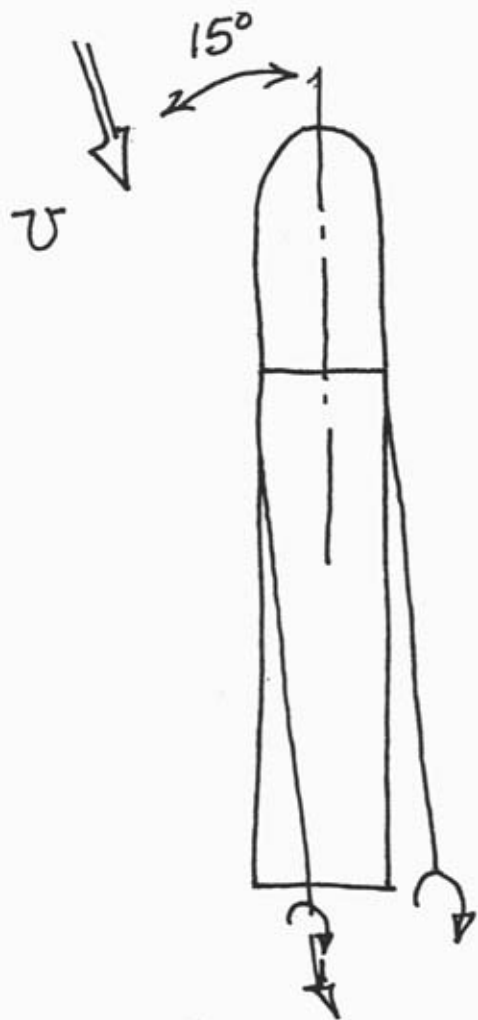
- ▲ SAME QUALITATIVE FLOW PICTURE $4^\circ - 6^\circ$ YAW
- ▲ SHARP JUMP AT GAP_c TO SEVERE WINDWARD/LEEWARD GAP FLOW
- ▲ HYSTERESIS $O(.05\sqrt{A})$ FOR GAP INCREASING / GAP DECREASING
- ▲ CRITICAL GAP DECREASES WITH INCREASING YAW ANGLE
 $.57\sqrt{A}$ AT 4° YAW \rightarrow $.53\sqrt{A}$ AT 6° YAW



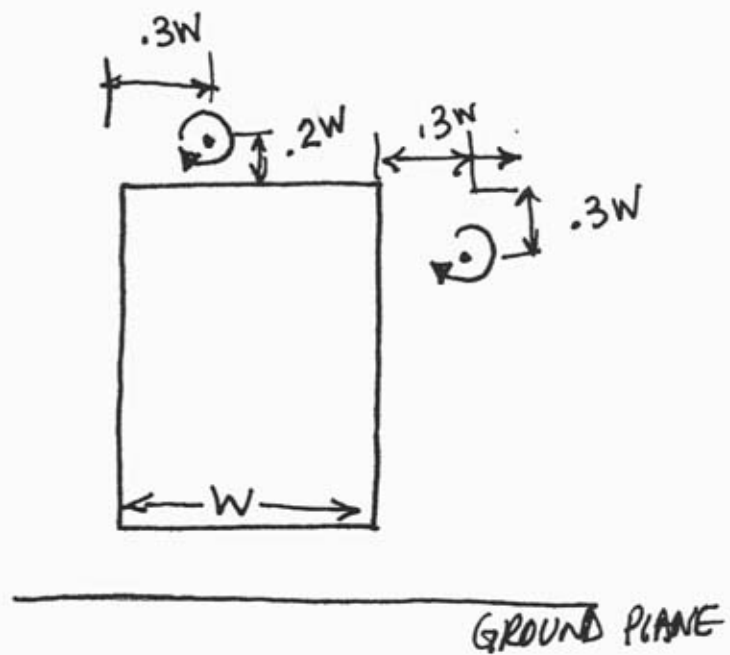
- ▲ SAME QUALITATIVE FLOW PICTURE 8° YAW & BEYOND
- ▲ JUMP OCCURS AT CRITICAL GAP, $GAP_c = .48\sqrt{A}$, 8° YAW
- ▲ GAP FLOW APPEARS LESS VIOLENT



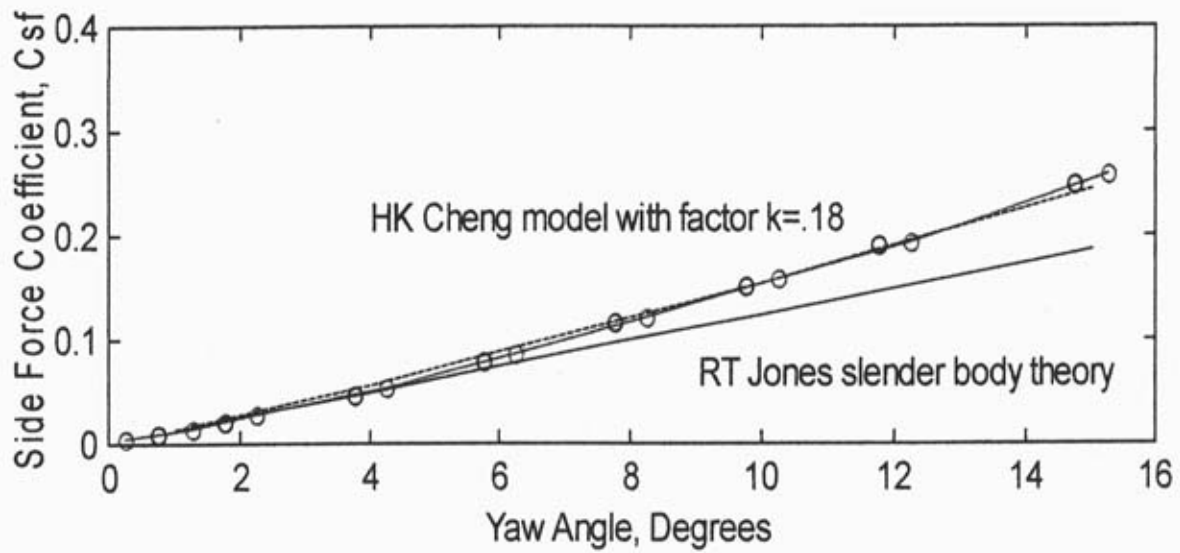
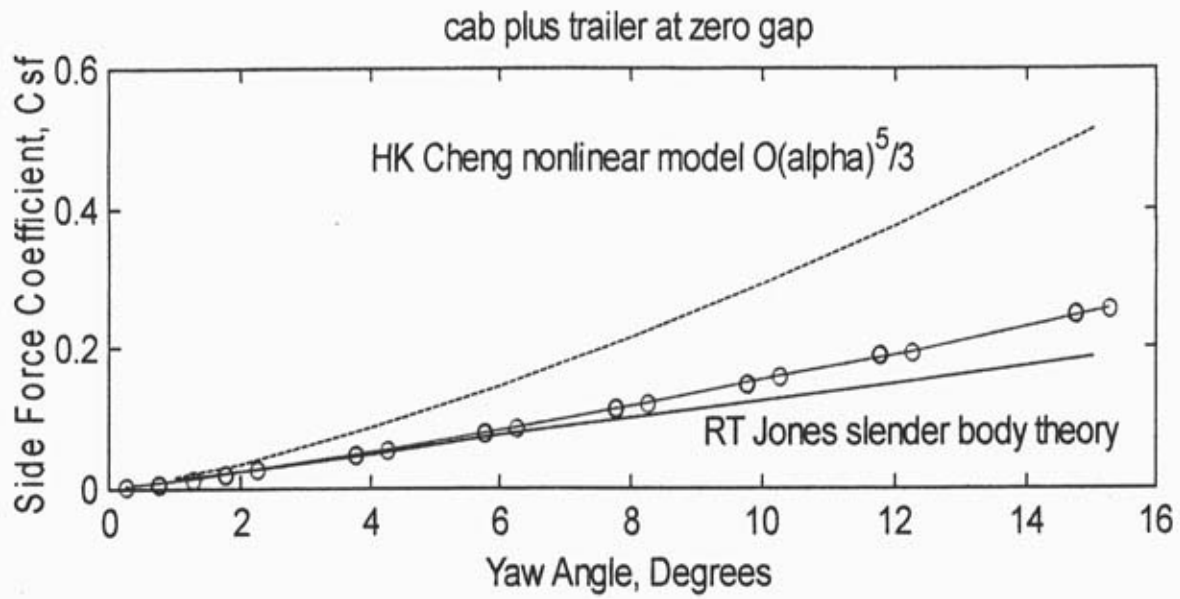




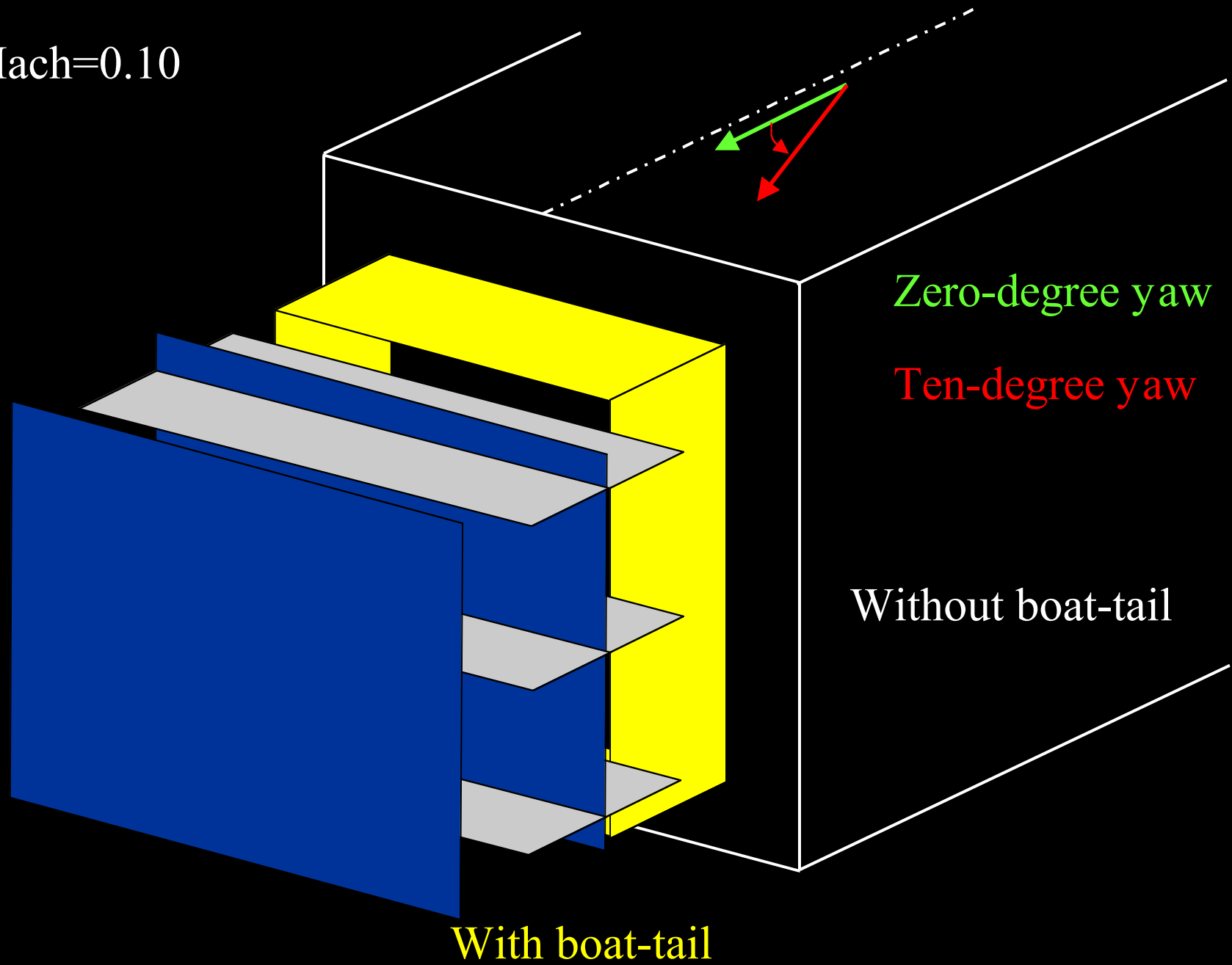
TDP VIEW



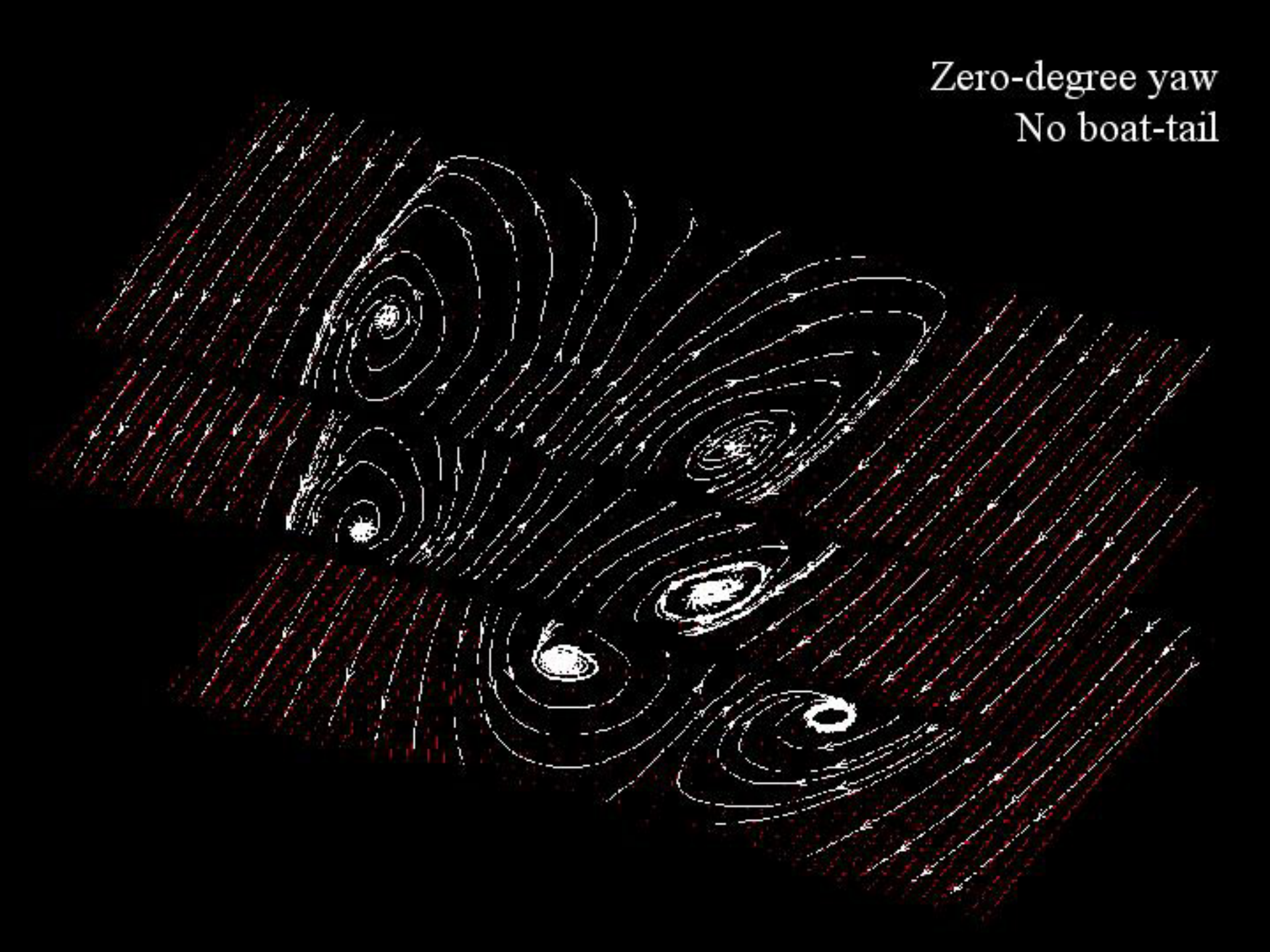
REAR VIEW



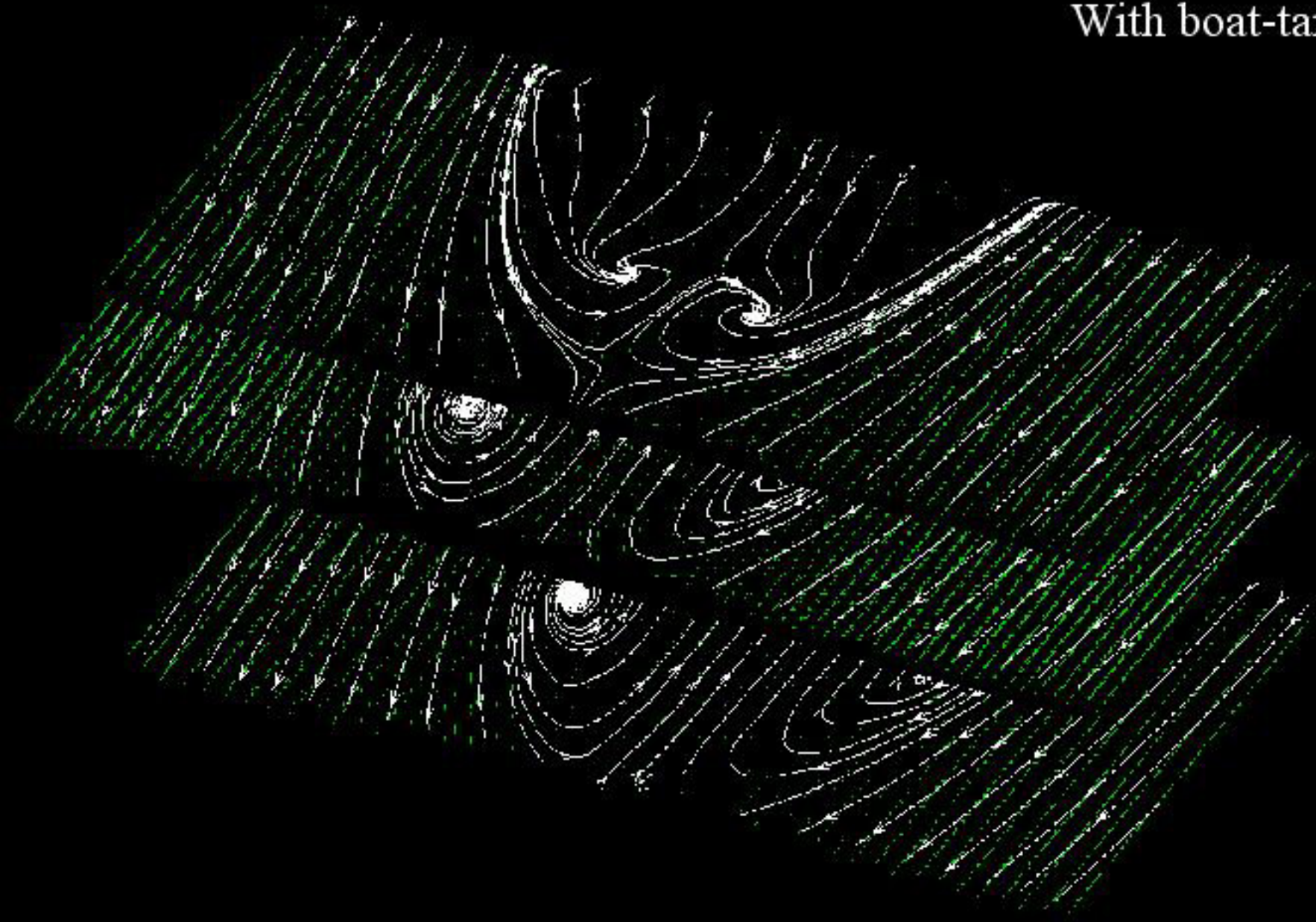
Mach=0.10



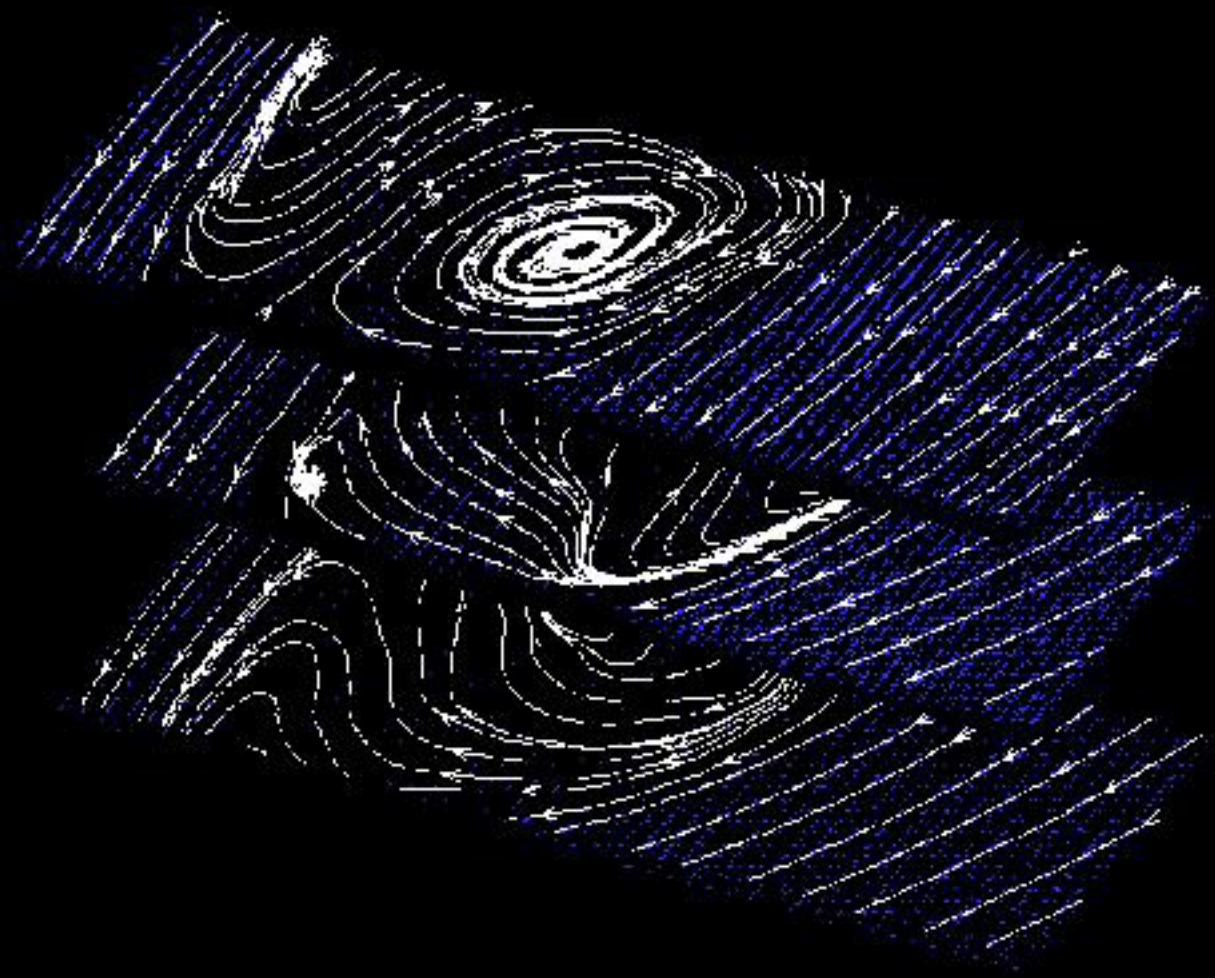
Zero-degree yaw
No boat-tail



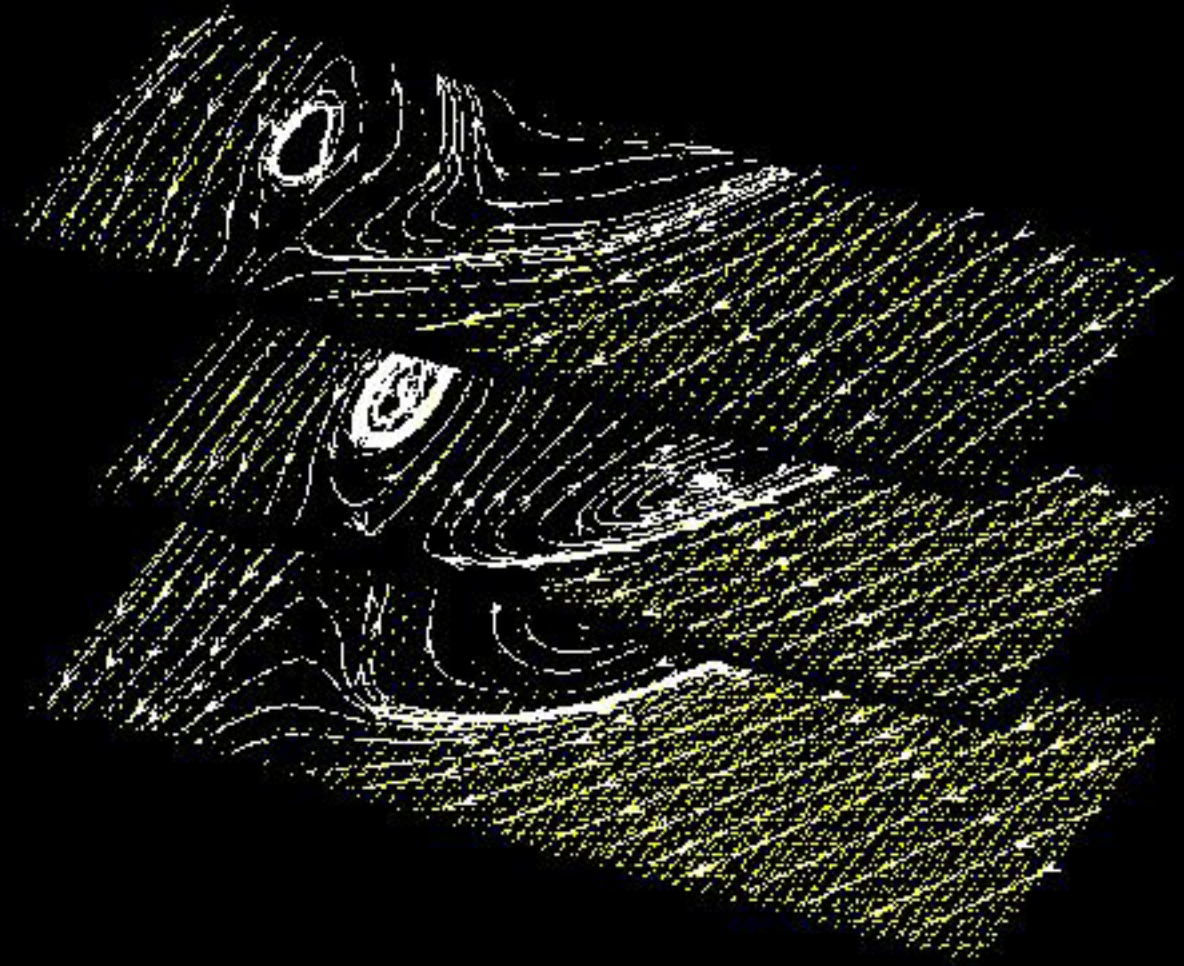
Zero-degree yaw
With boat-tail



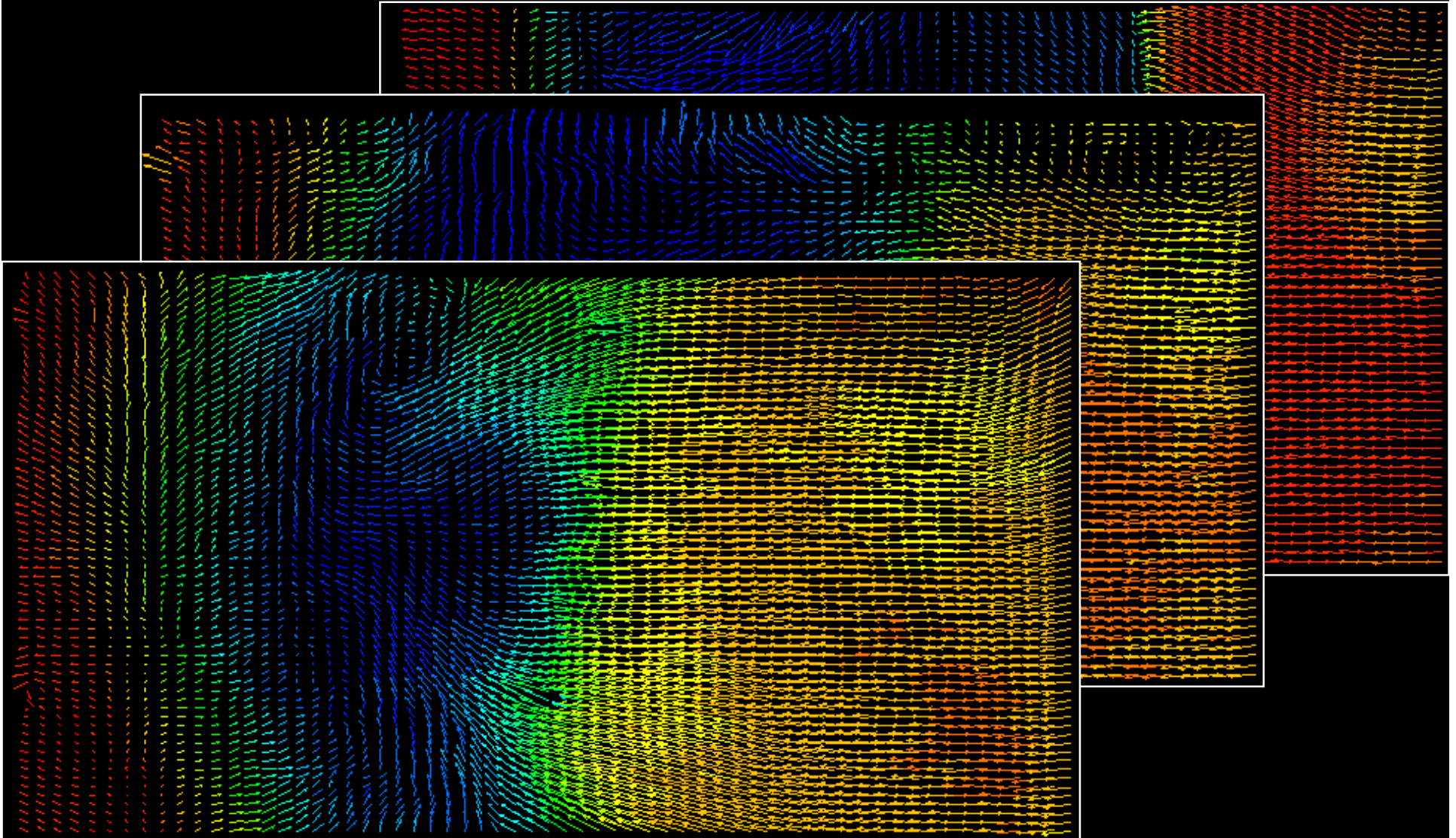
Ten-degree yaw
No boat-tail



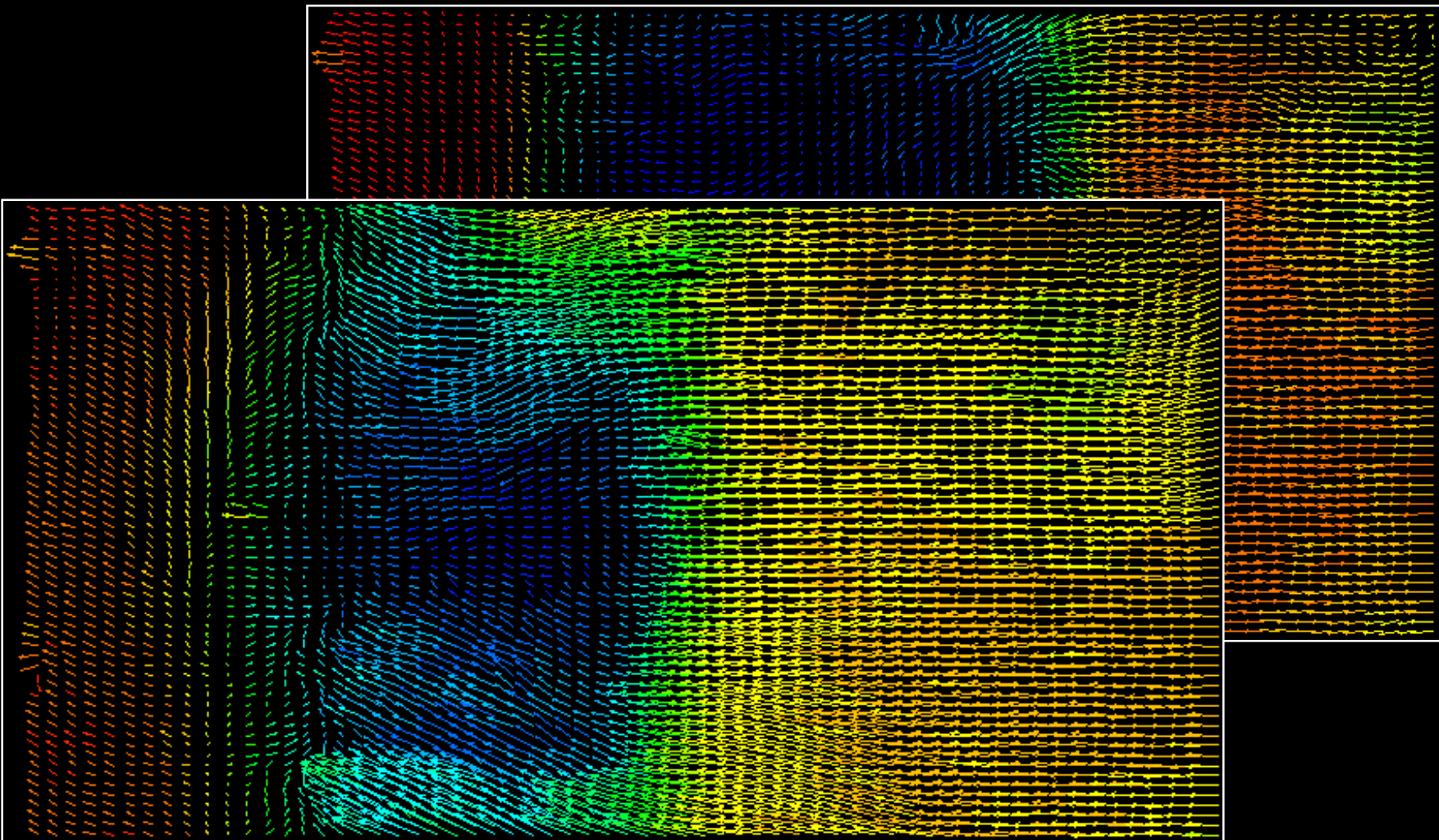
Ten-degree yaw
With boat-tail



Ten-degree yaw
No boat-tail



Ten-degree yaw
With boat-tail



Future Plans

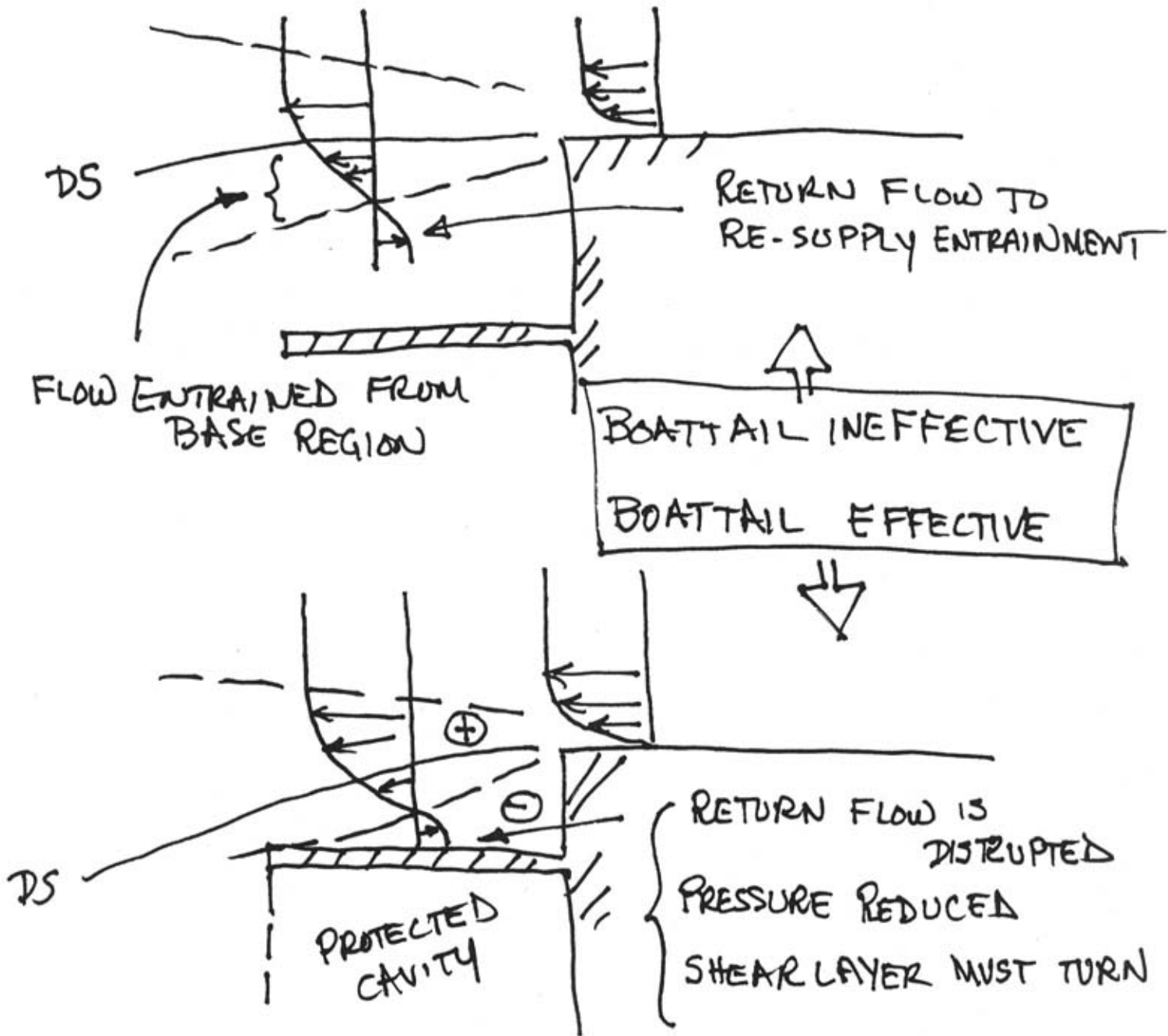
▲ Truck with gap & yaw

- Complete DPIV measurements
- Data analysis
- Preparation of comprehensive paper

▲ Comprehensive comparison between RANS LES and NASA tests

▲ ? Provide additional DPIV in our wind tunnel to complement existing data on boattail plates

How does boattail plate work?



QUESTIONS:

WHAT ARE LIMITS OF EFFECTIVENESS?
WHAT IS ROLE OF PROTECTED CAVITY?



Computational Prediction for a Simplified Truck Geometry

Kambiz Salari

Walter H. Rutledge

**Aerosciences and Compressible Fluid Mechanics Department
Sandia National Laboratories**

**Heavy Vehicle Aerodynamic Drag: Working Group Meeting
Lawrence Livermore National Laboratory**

August 15, 2000



Outline

- **Tasks completed for FY00**
- **Feasibility of RANS approach for base flow predictions**
- **DES effort**
- **Conclusions**



Sandia FY00 Tasks

Initial conditions for NASA 7x10	Done
New viscous grids for 7 yaw angles including the boattail plates	Done
Flow calculations for up to 7 yaw angles	Done, 0° & 10°
Flow calculations with boattail plates	Not done
Grid resolution studies	Not done
Comparison with NASA exp. data	Done
Document RANS solutions in SAE paper	In progress, March 2001
DES research	In progress
Future NASA wind tunnel experiments	Unknown



NASA 7'x10' Tunnel, Flow Simulation

Test Condition for run 7:

Total pressure = 102,652.76 (N/m²)

Static pressure = 97,612.51 (N/m²)*

Dynamic pressure (Q) = 5,040.24 (N/m²)

Static Temperature = 5° C

Mach number = 0.27

Yaw angle = 0° and 10°

Re = 2x10⁶ (based on truck width)

*Based on equivalent "Tunnel Empty" Condition



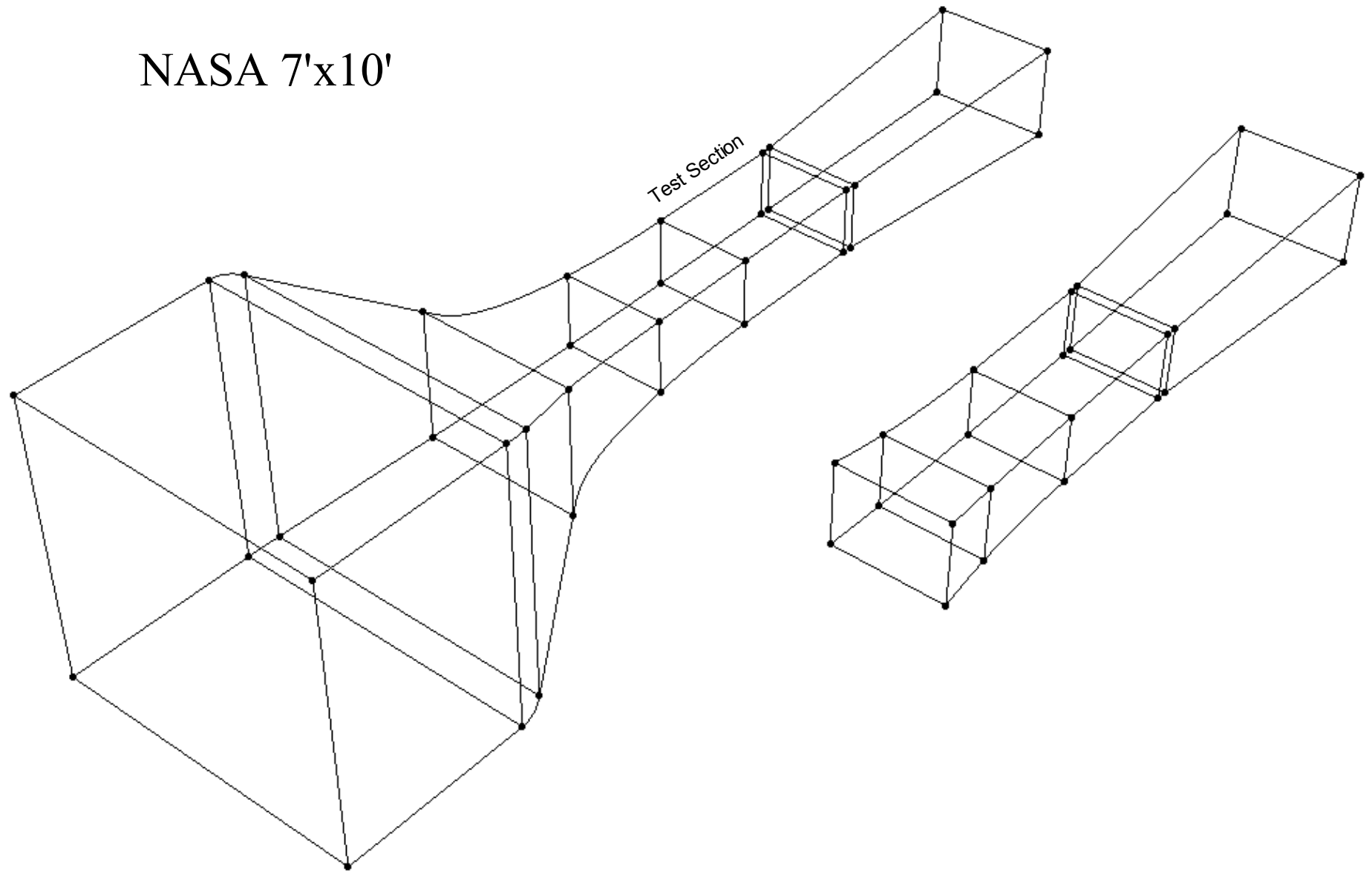
SACCARA Code Capabilities

Sandia Advanced Code for Compressible Aerothermodynamics Research and Analysis

- Multi-block, structured grids for 2-D, Axisymmetric, and 3-D flows
- Solution of the Full Navier-Stokes equations for compressible Flows
- Finite volume spatial discretization (steady and unsteady)
- MP implementation on a variety of distributed parallel architectures (IBM, Intel, etc.)
- Implicit time advancement schemes
- Subsonic → Hypersonic flows
- Zero-, one-, and two-equation turbulence models
- Ideal, equilibrium, and thermo-chemical nonequilibrium finite-rate gas chemistry
- Ablation boundary conditions
- Rotating coordinate system



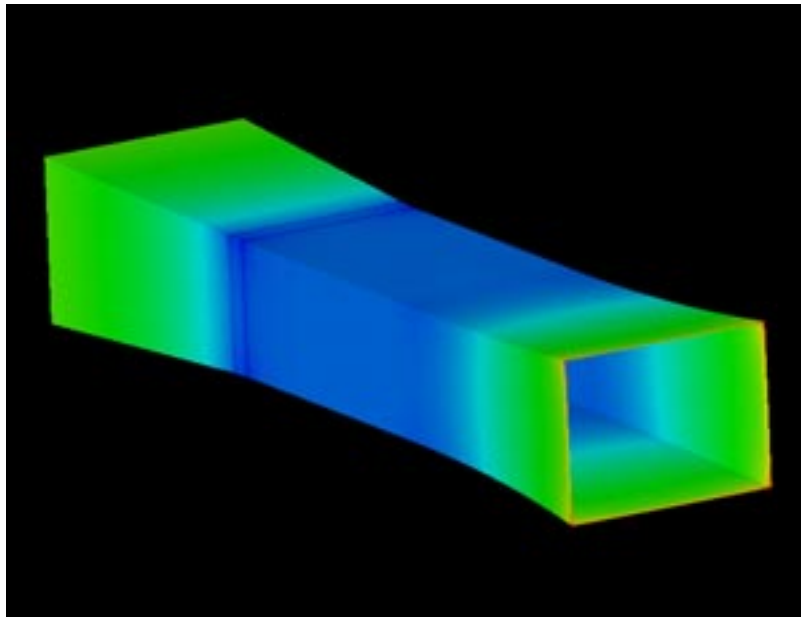
NASA 7'x10'



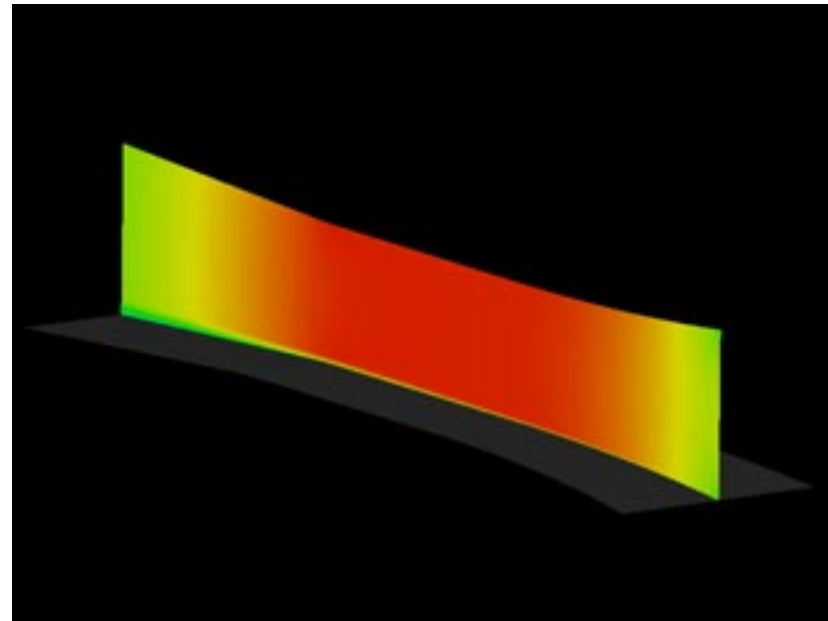


Flow Simulation of NASA 7'x10' Tunnel

Pressure contours

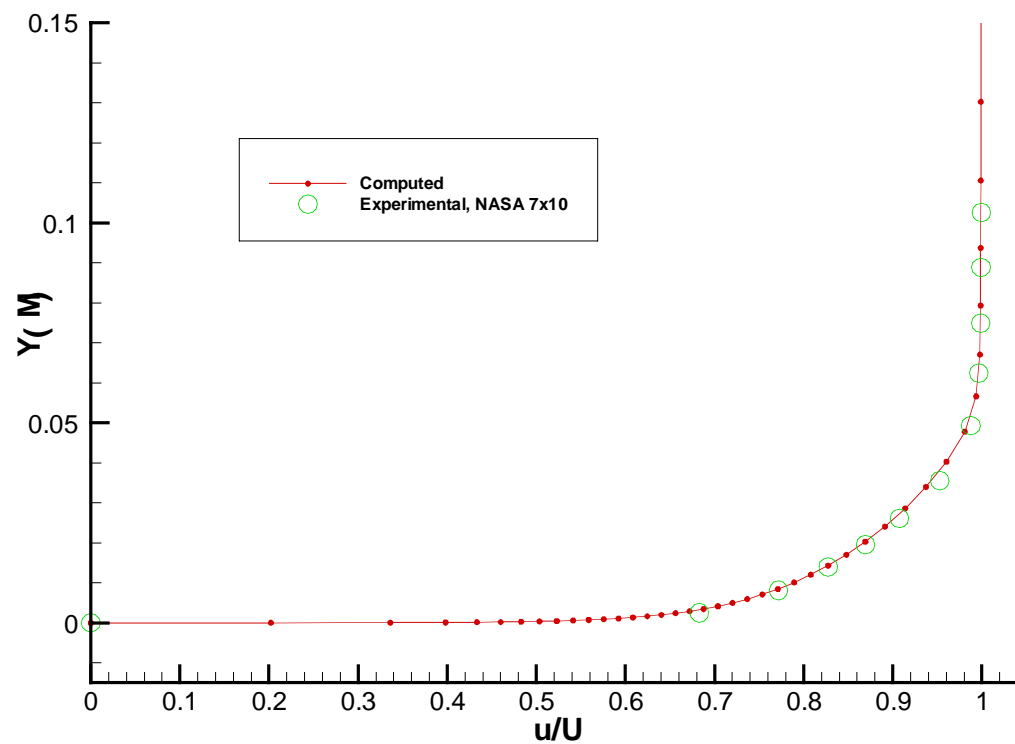


Mach contours (centerline)

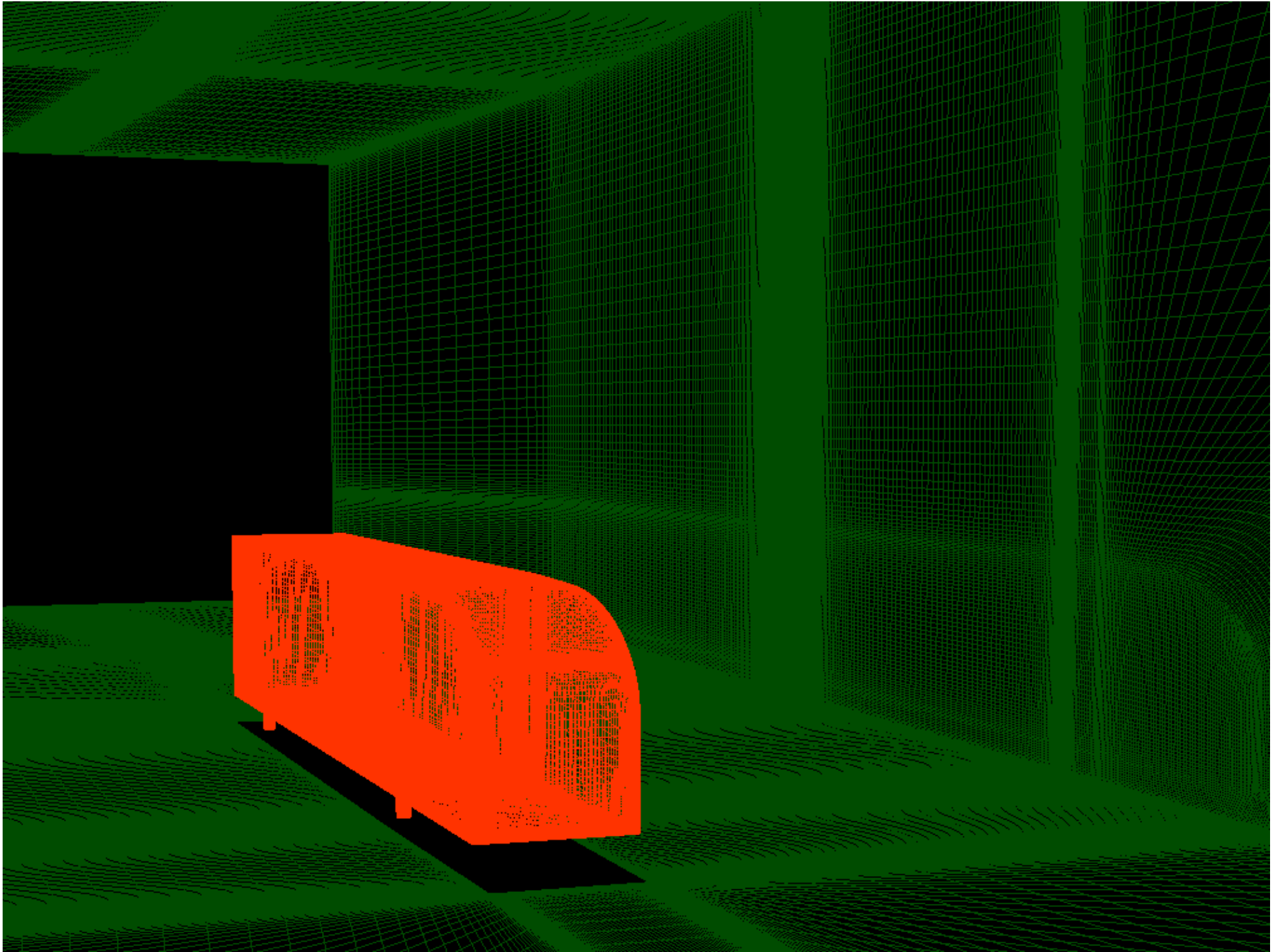




Inflow Velocity Profile (Test Section)



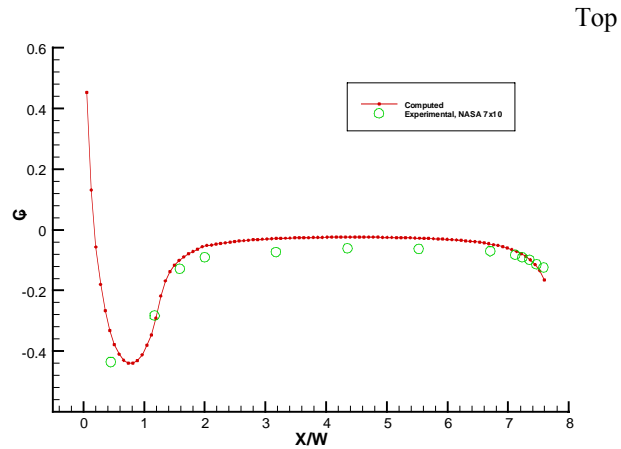




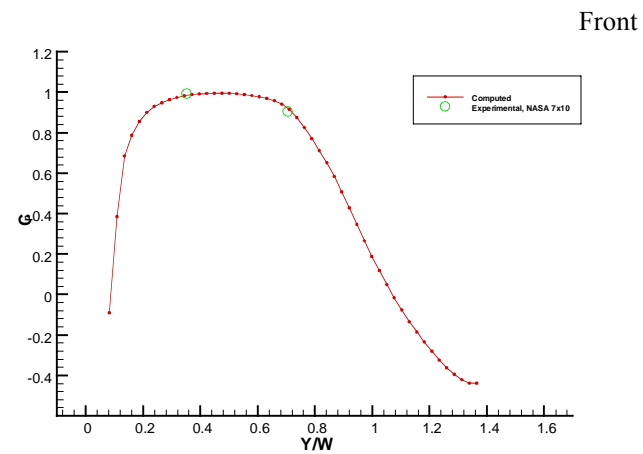
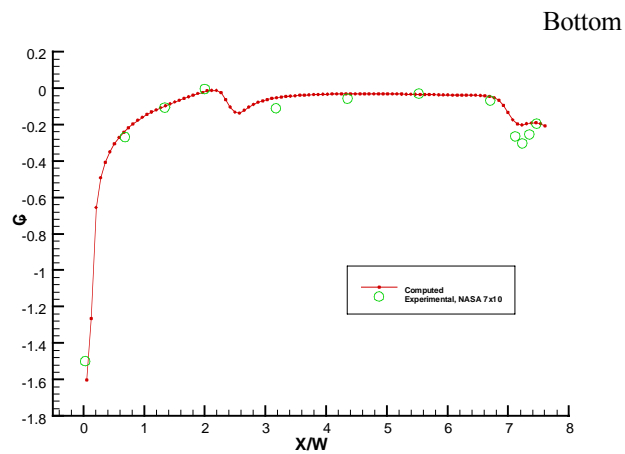
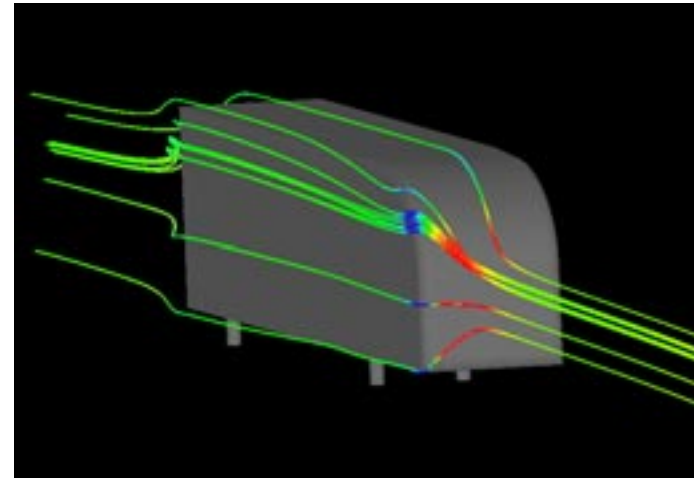


Steady RANS Solution, 0° yaw

Surface C_p comparison to NASA data (model centerline)



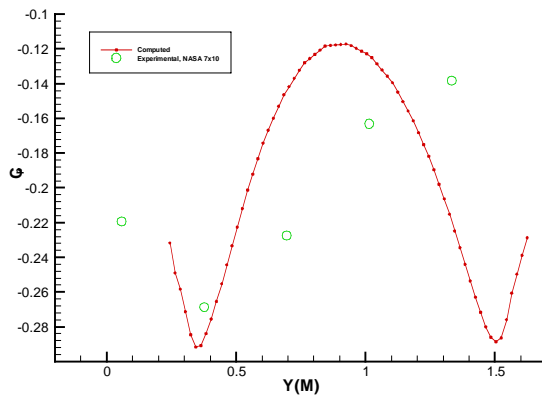
Particle traces are colored by pressure magnitude



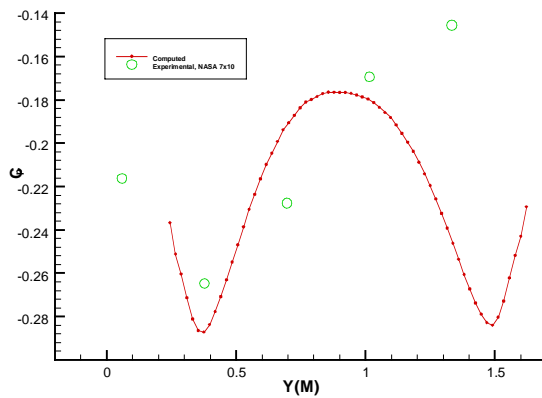


Steady RANS Solution, 0° yaw, ...

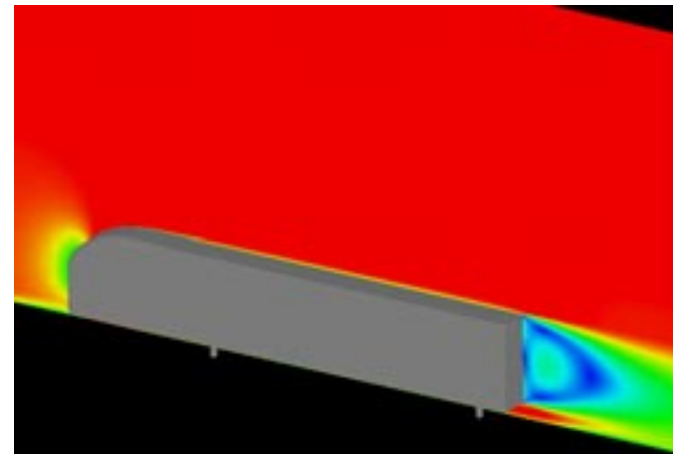
$Z = 0.0$ (centerline)



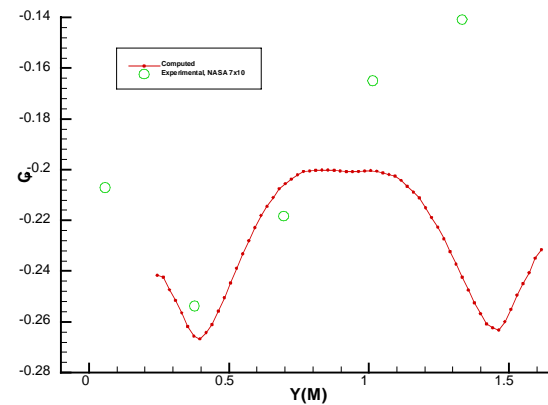
$Z = 0.22$



Mach contours

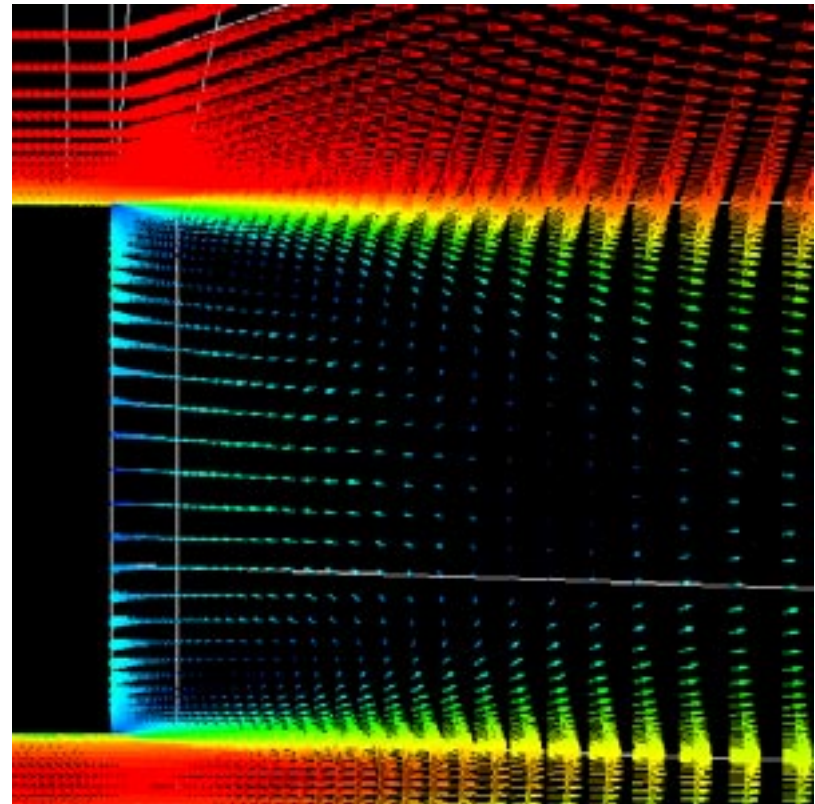
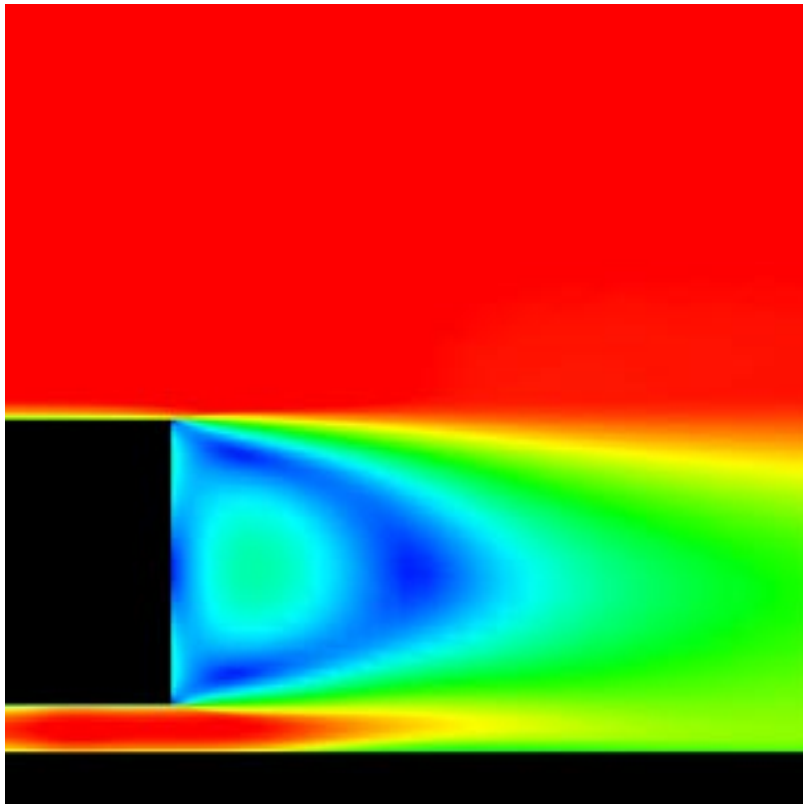


$Z = 0.44$



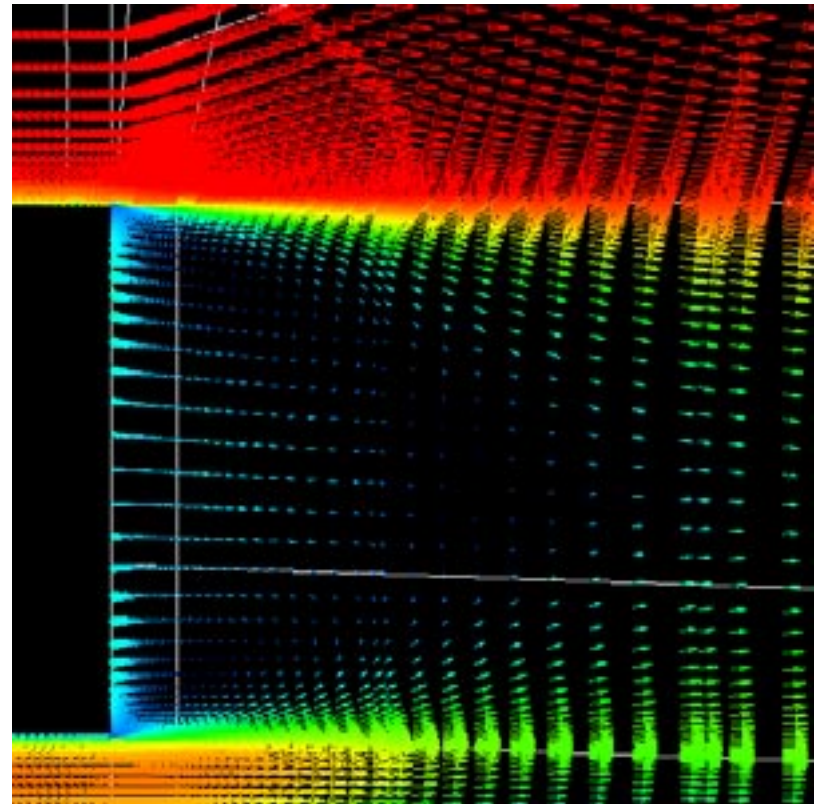
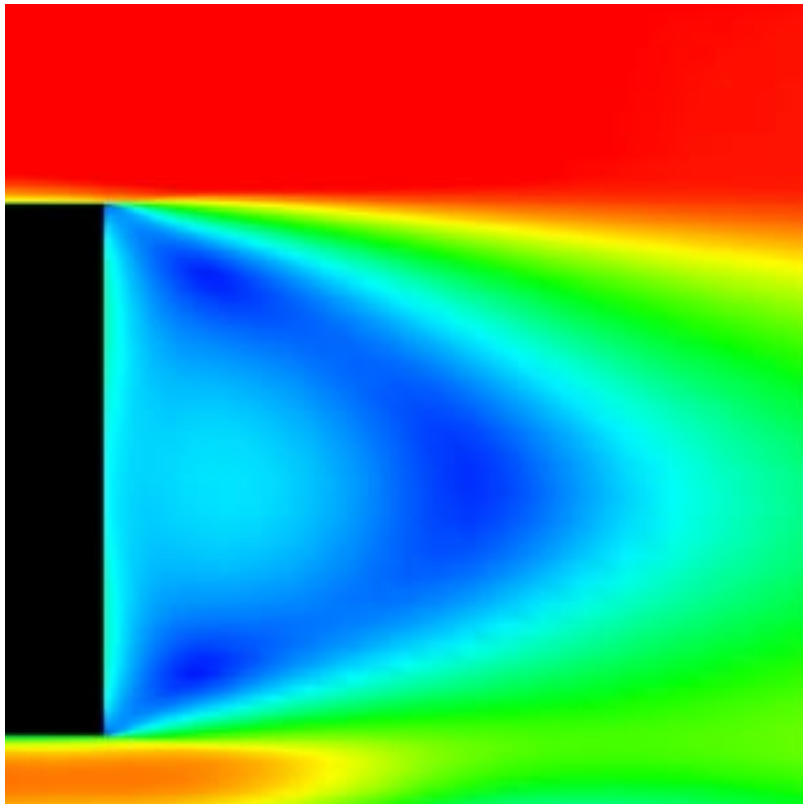


Steady RANS Results , 0° yaw, ...



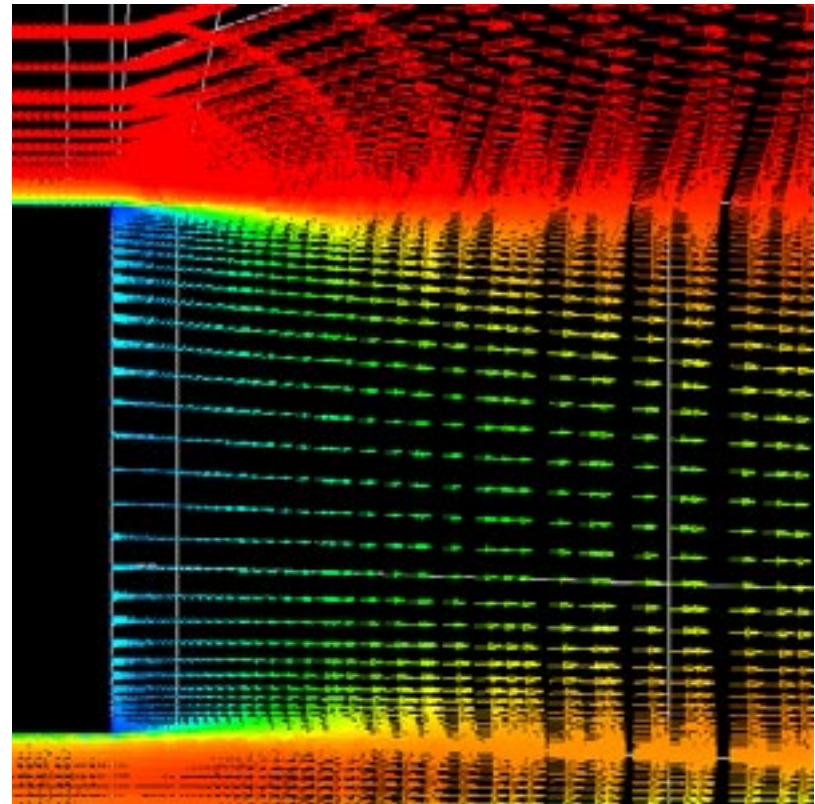
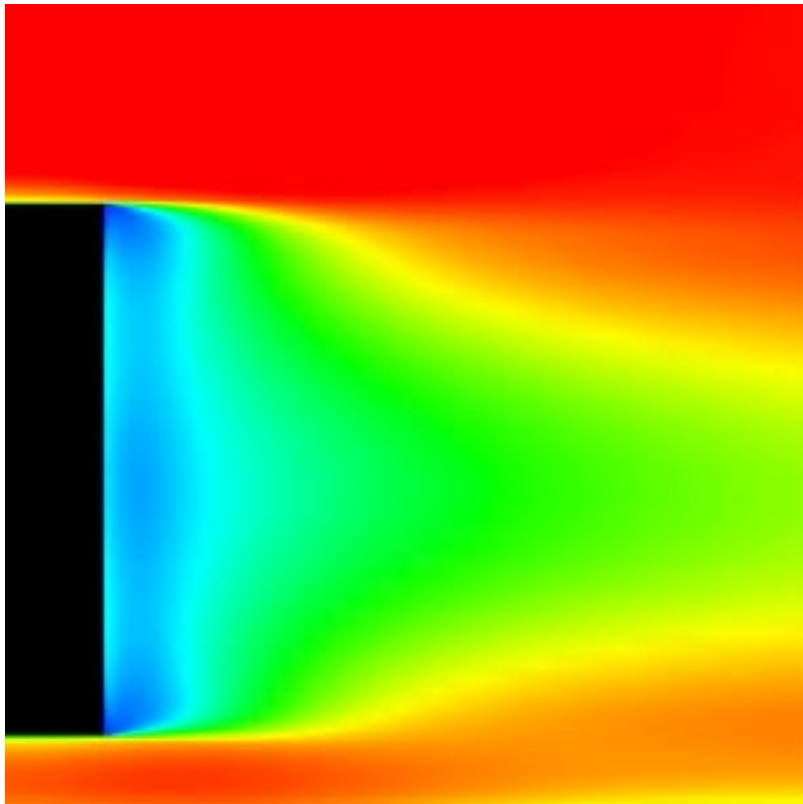


Steady RANS Results , 0° yaw, ...





Steady RANS Results , 0° yaw, ...





Turbulence Modeling Approaches

- Reynolds-Averaged Navier-Stokes Simulations (RANS)
- Detached Eddy Simulations (DES)
 - LES type calculation with RANS wall boundary conditions
- Large Eddy Simulations (LES)
- Direct Numerical Simulations (DNS)



Subgrid Scale Stress Model for DES

The unsteady form of the Spalart-Allmaras RANS one-equations turbulence model is used to provide the eddy viscosity **in the boundary layer**

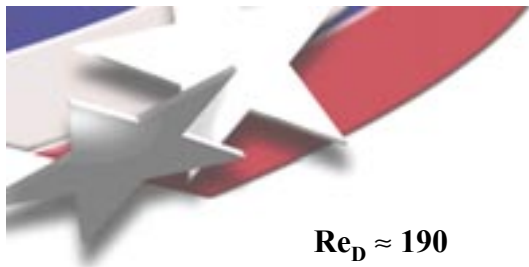
The same model needs to be modified to the appropriate eddy viscosity for LES **outside the boundary layer**



Problem Selection for DES Simulations

The flow over a **circular cylinder** at $Re_D=3900$ is selected because

- Geometric simplicity
- Complex flow features
- Extensively studied both numerically (2D & 3D) and experimentally
- Large number of review articles



$Re_D \approx 190$

$Re_D \approx 190 - 260$

$Re_D \approx 260$

$Re_D \approx 1000 - 200,000$

Primary Karman
vortex shedding

Small scales

Large scales

Mode A
vortex loops

Mode B
streamwise
vortex pairs

Vortex Dislocations

Large-scale 3-D distortions

Axial core flows

Shear layer instabilities
at higher
Reynolds numbers



Test Cases and Meshing

Two-Dimensional Calculations

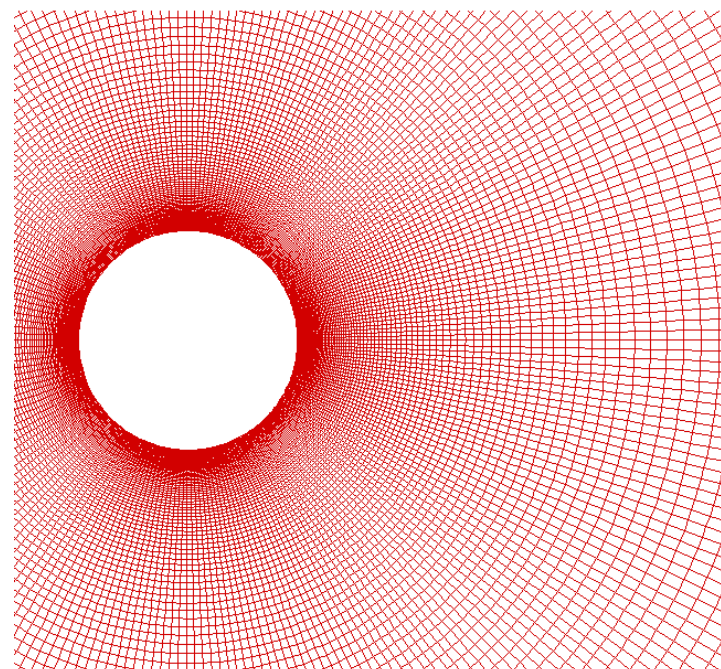
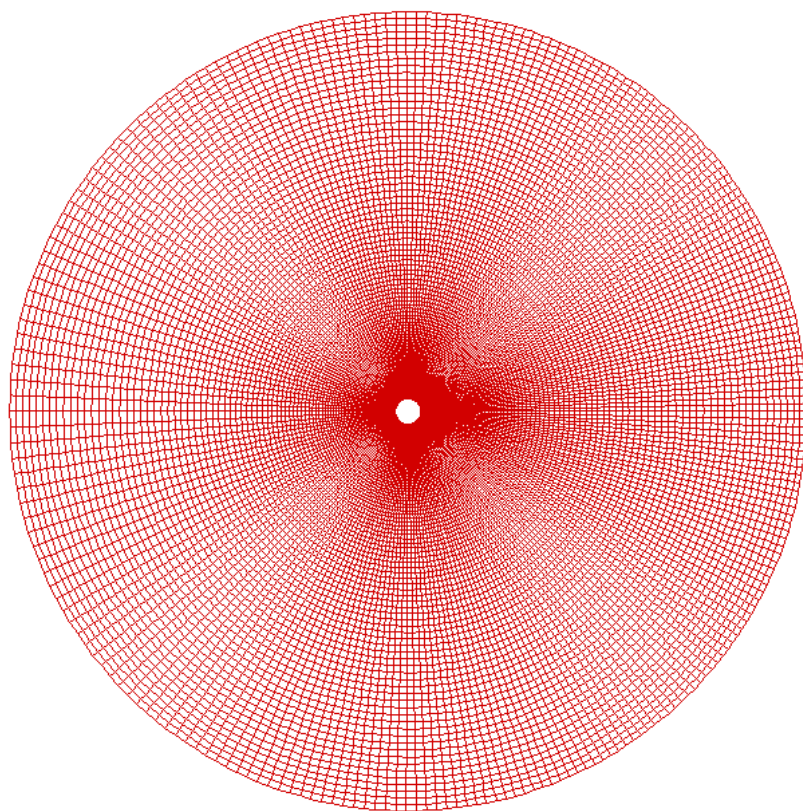
- **Steady RANS**
- **DES**
- **O-type meshes (121x81, 241x161, 321x281)**

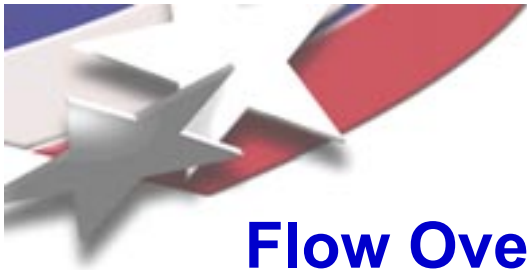
Three-Dimensional Calculations

- **DES**
- **The grids are constructed with sweeping the 2D O-type mesh in the third direction (121x81x21, 241x161x41, 321x281x81)**



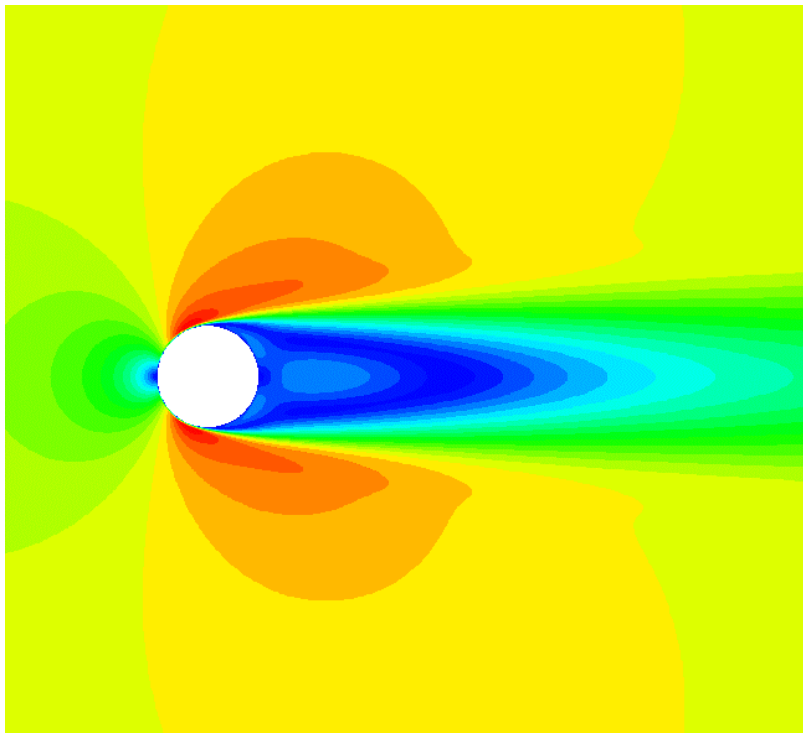
O-Type Mesh, 241x81



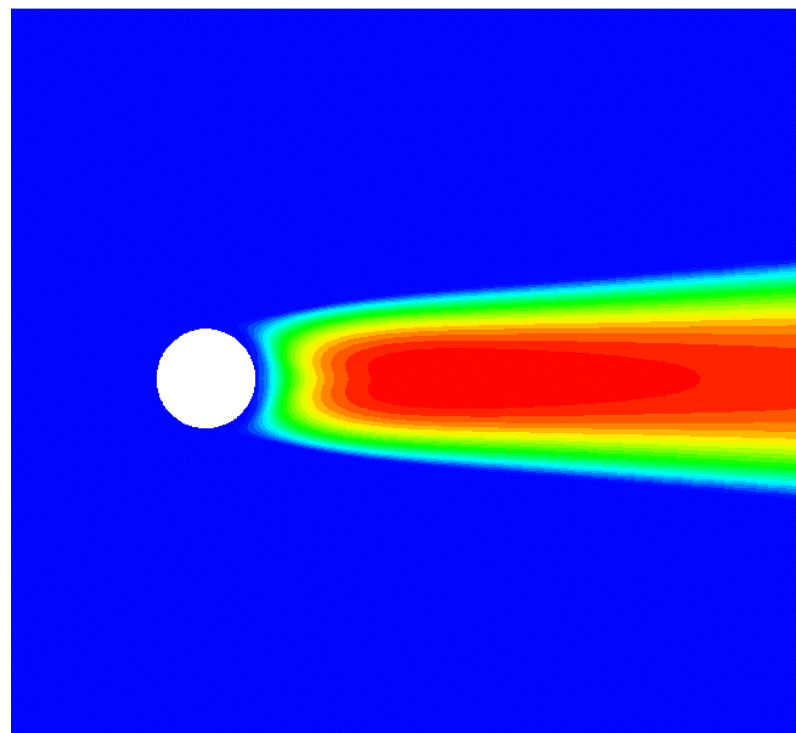


Flow Over a Circular Cylinder, 2D Steady RANS

Mach

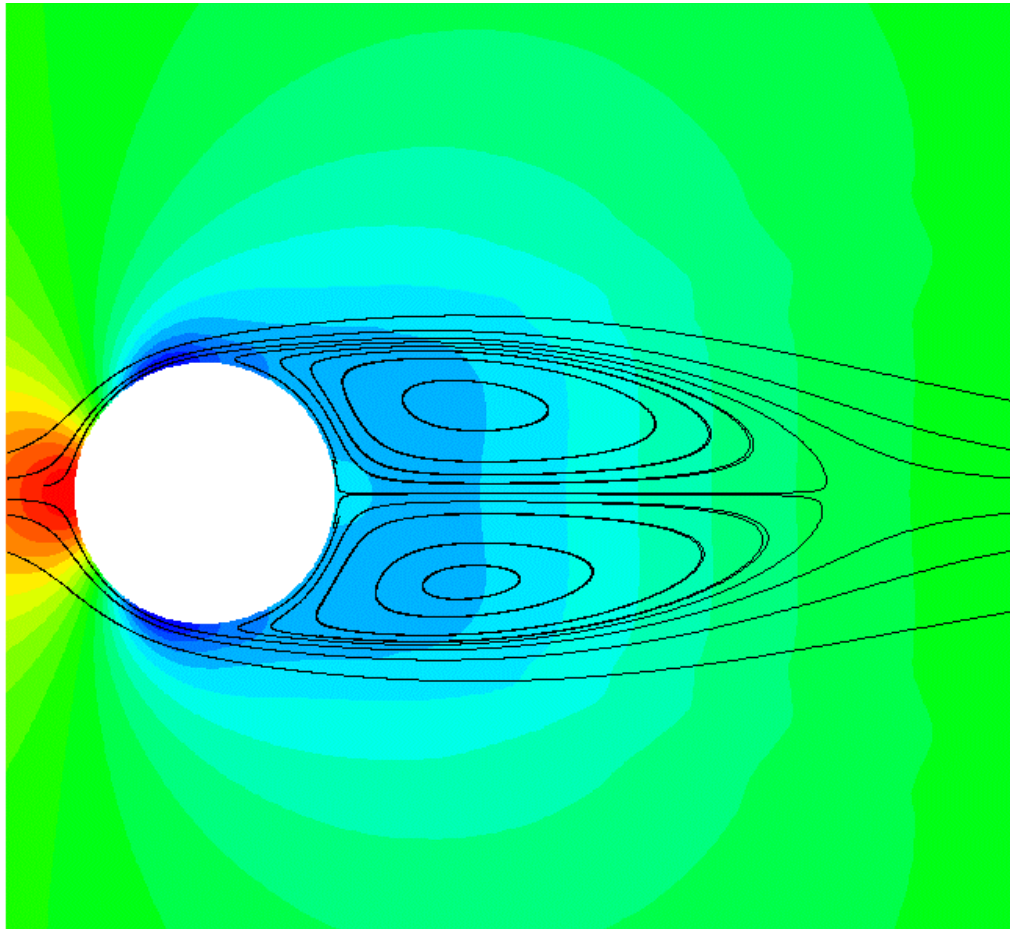


Turbulent Viscosity





Flow Over a Circular Cylinder, 2D Steady RANS, ...





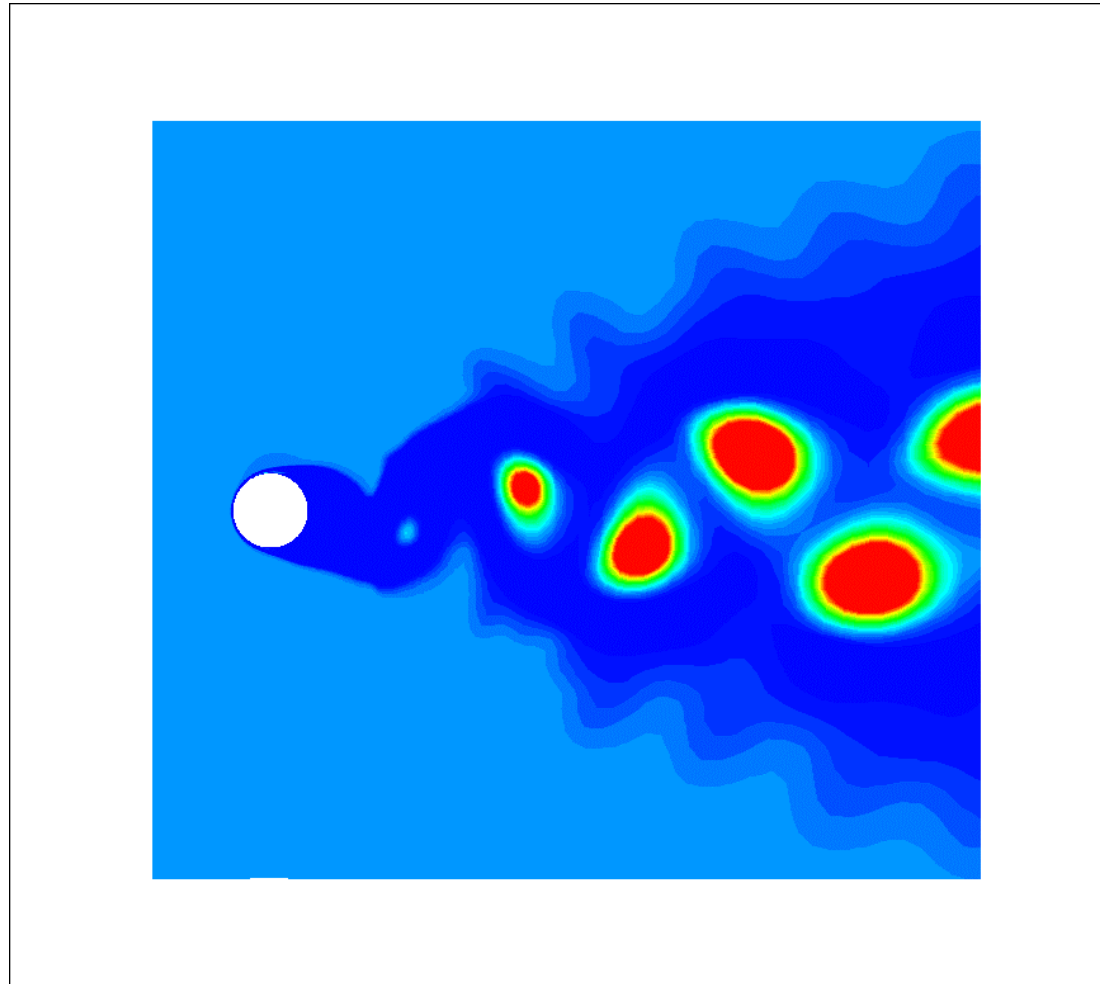
Flow Over a Circular Cylinder, 2D Steady RANS, ...

Grid Size	C_d	$-C_{pb}$	θ_{sp}	L_{red}/D	U_{min}
121x81	0.90	0.536	92.4°	1.43	-0.31
241x161	0.89	0.609	89.7°	1.85	-0.28
321x281	0.86	0.594	89.6°	1.90	-0.25
Exp.	0.99±0.05	0.88±0.05	86.0°±2.0°	1.4±0.1	-0.24±0.1



Flow Over a Circular Cylinder, 2D DES Simulation

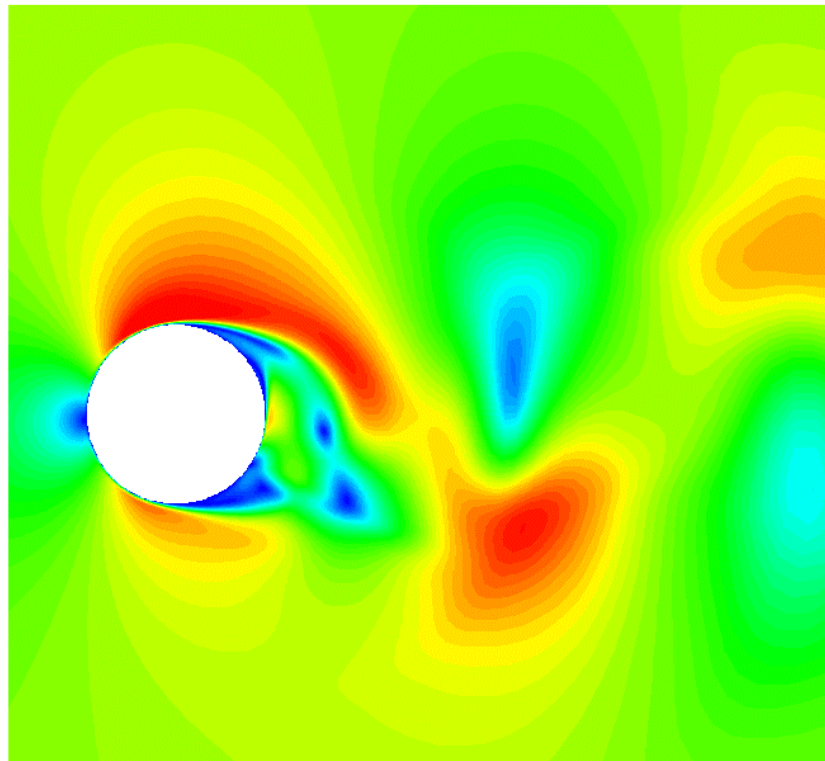
Contours of turbulent viscosity





Flow Over a Circular Cylinder, 2D DES Simulation

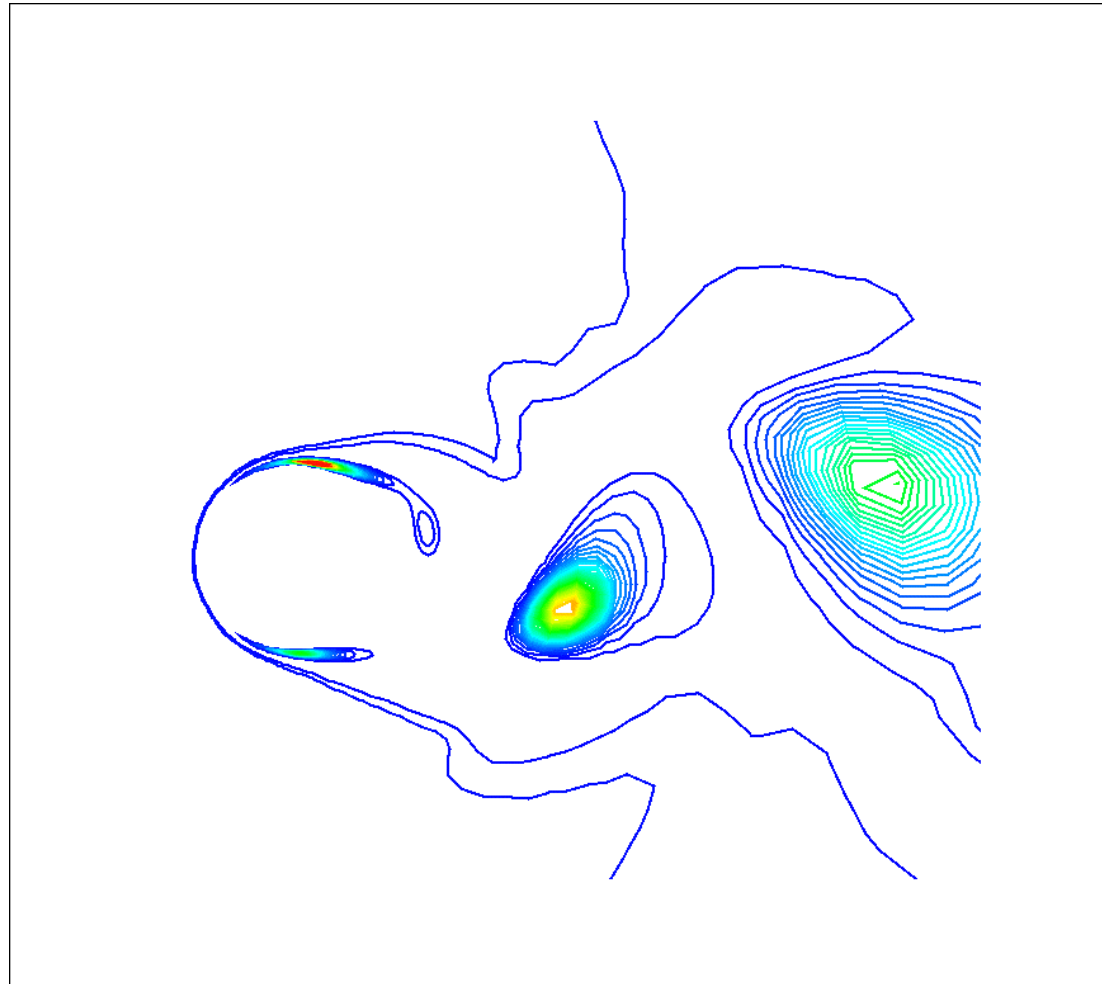
Contours of velocity magnitude





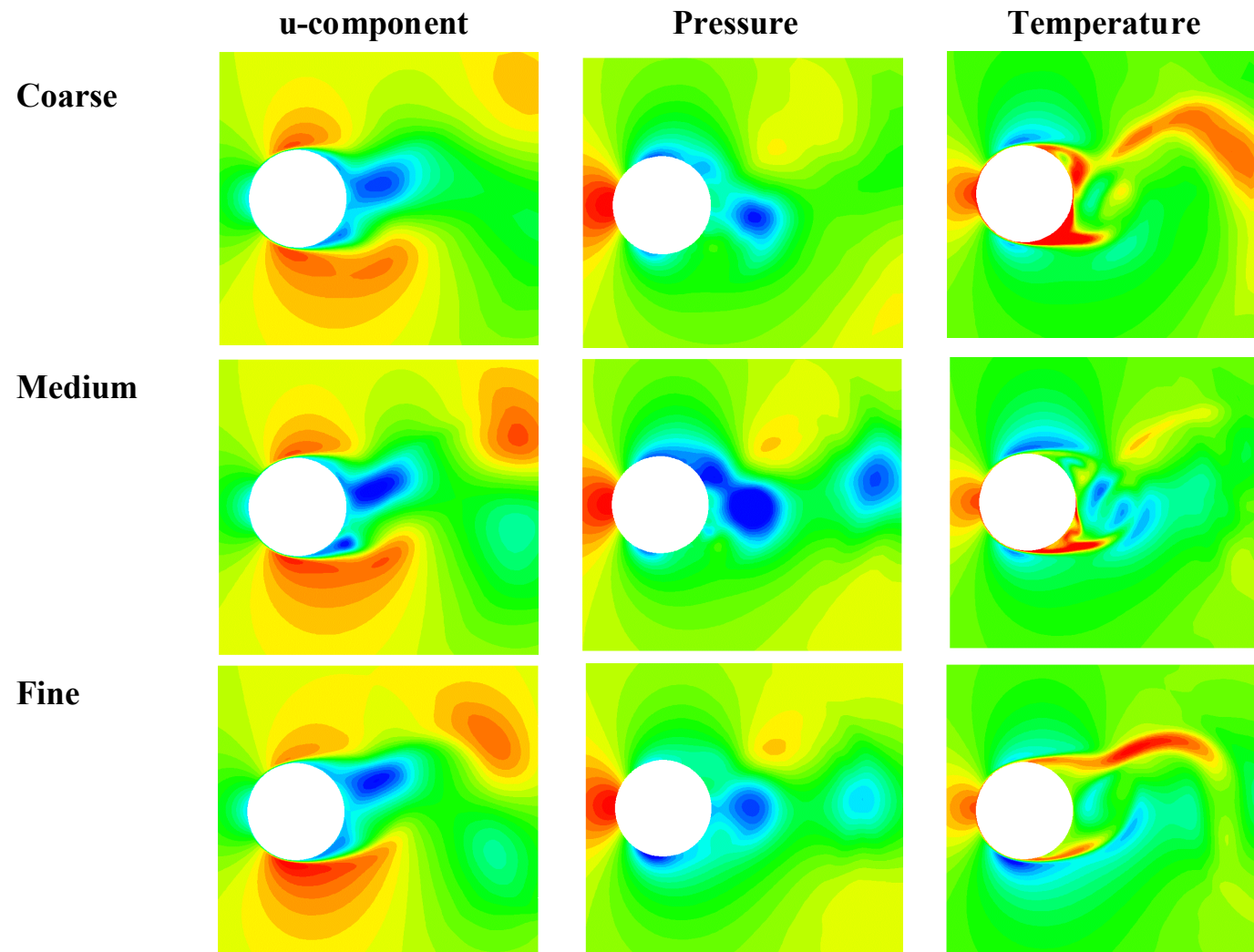
Flow Over a Circular Cylinder, 2D DES Simulation

Contours of turbulent viscosity



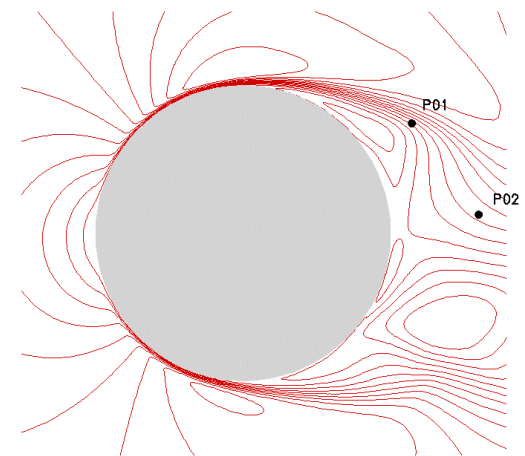
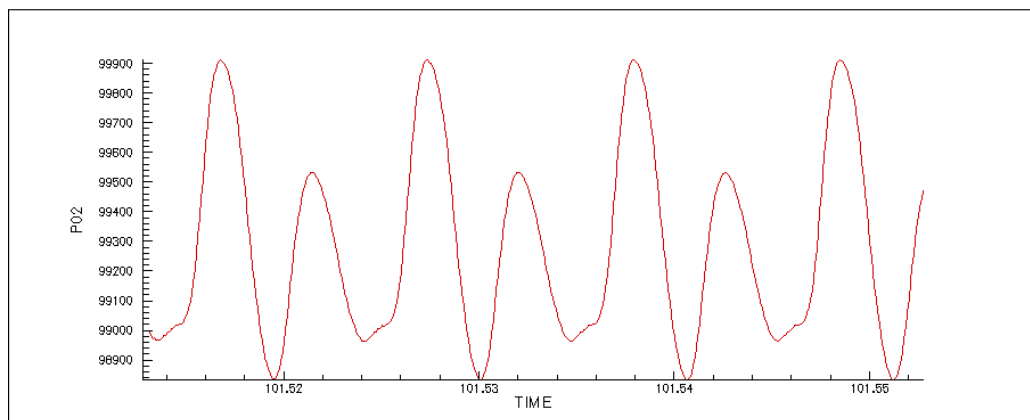
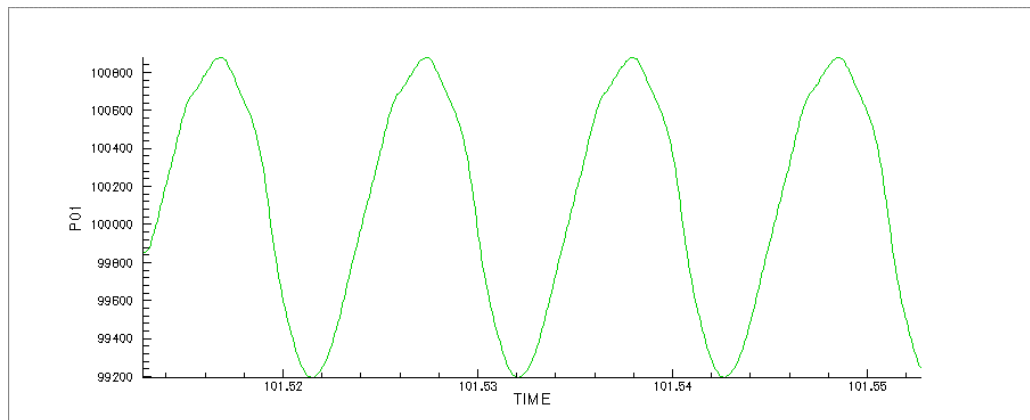


Grid Resolution Study for the 2D DES Simulations



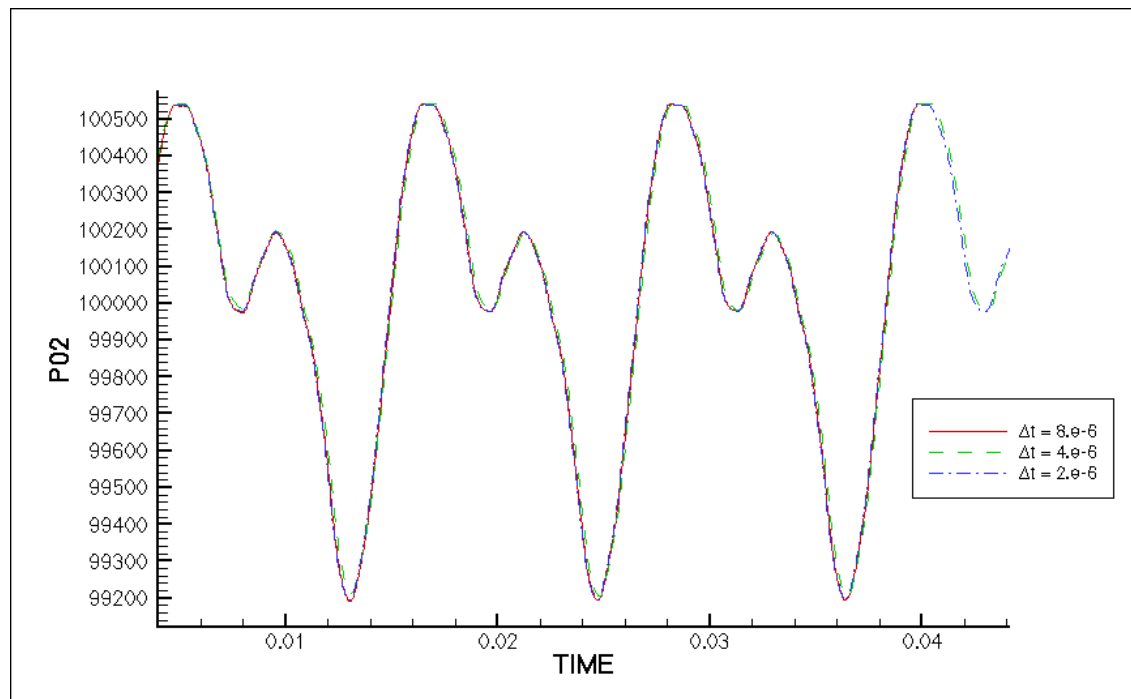


2D DES Simulation



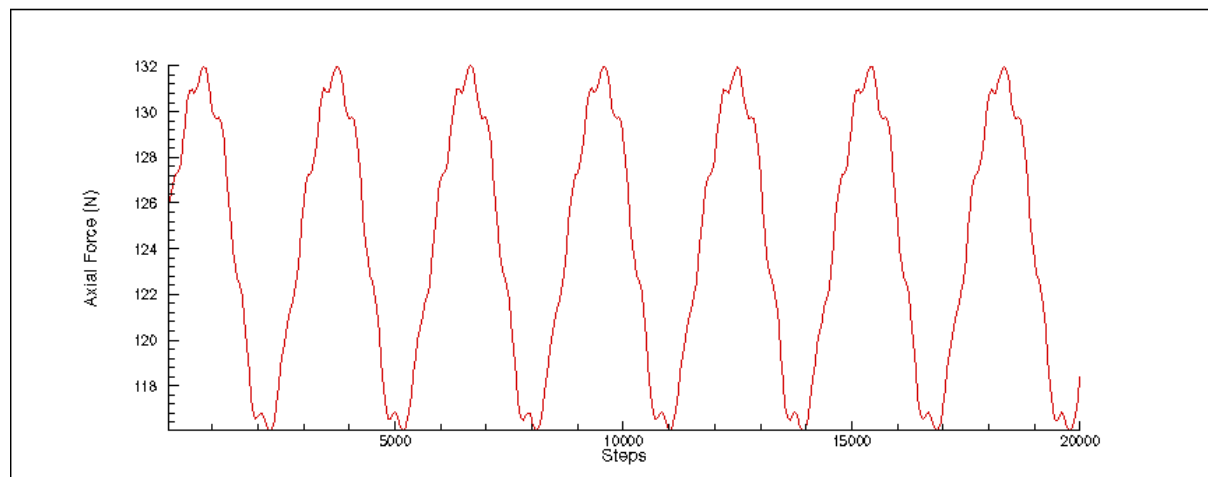
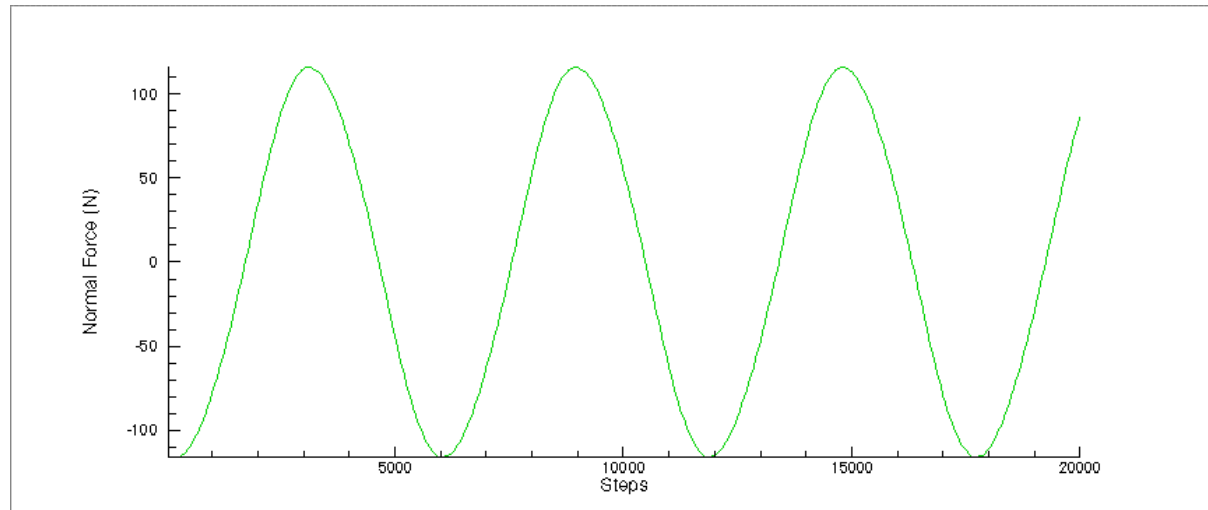


Time Resolution Study for the 2D DES simulations





Force Exerted on the Cylinder, 2D DES Simulations





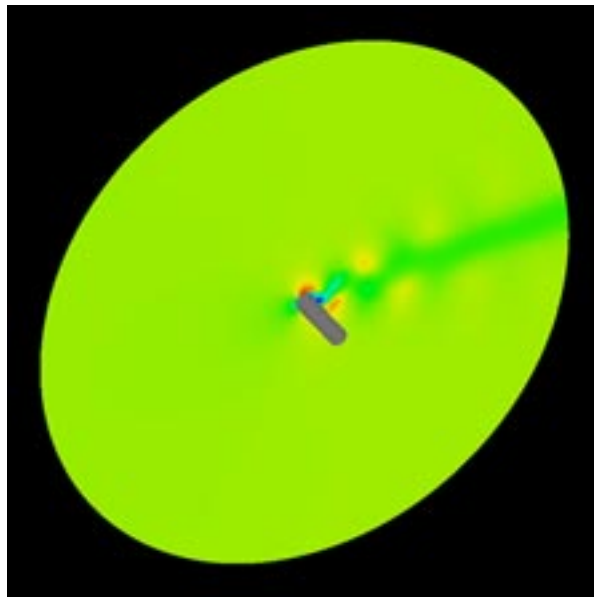
2D DES Results

Grid Size	C_d	St
121x81	1.30	0.214
241x161	1.49	0.237
321x281	1.34	0.238
Exp.	0.99 ± 0.05	0.215 ± 0.005

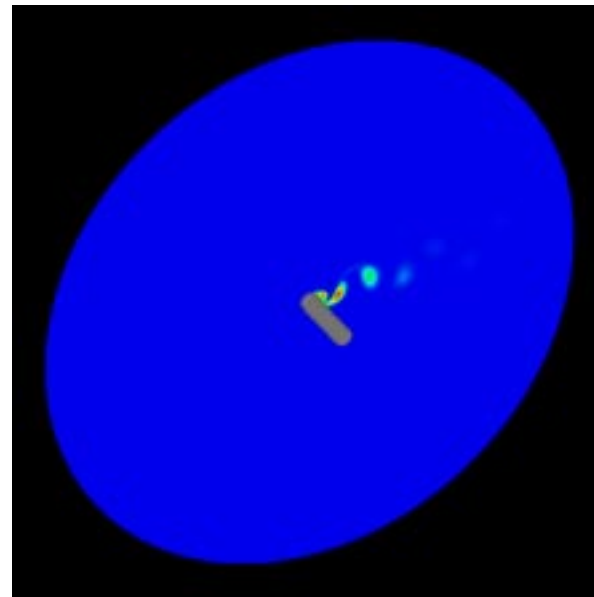


3D DES Simulation (Medium Grid)

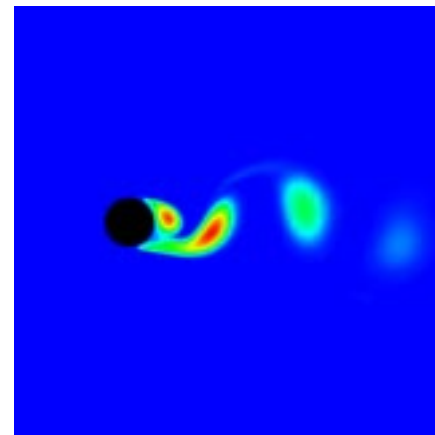
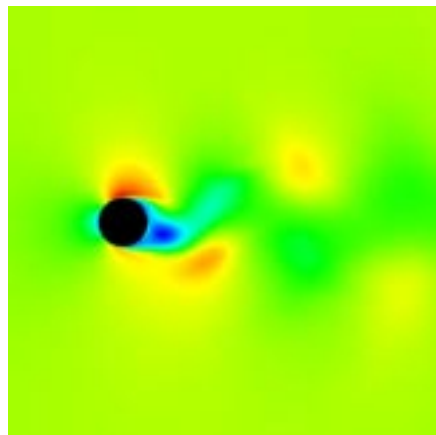
**u-component
of velocity**



**Turbulent
viscosity**



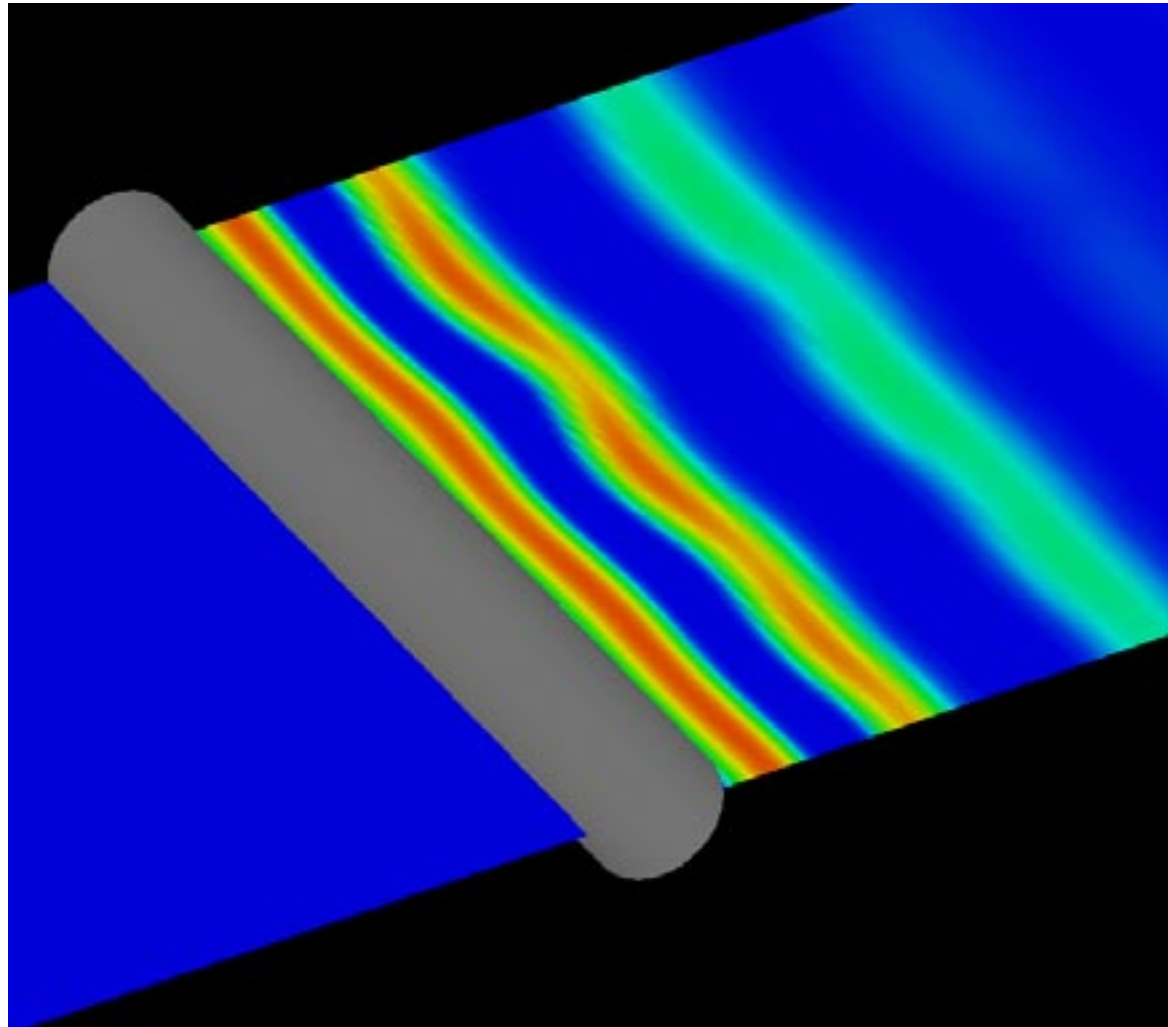
2D view





3D DES Simulation (Medium Grid) ...

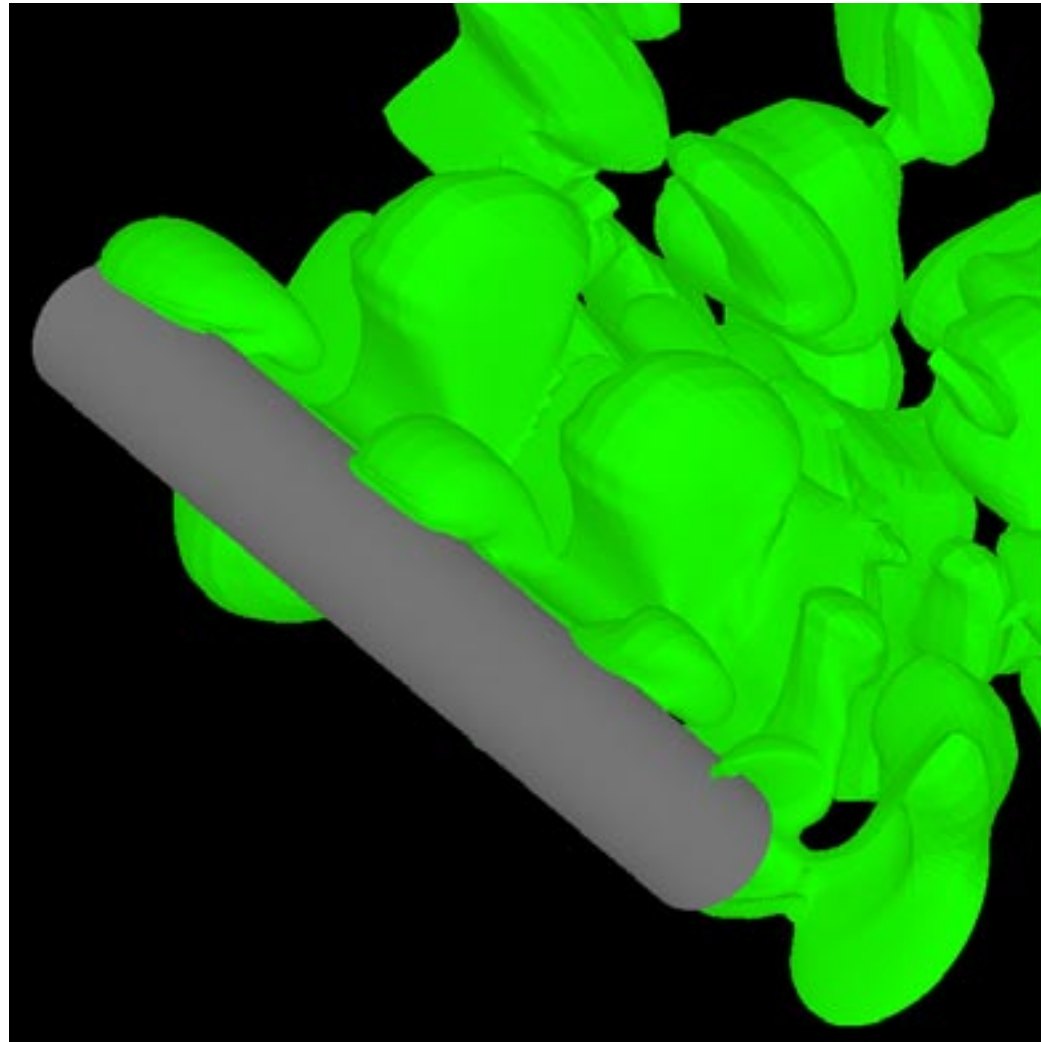
Turbulent viscosity
contours, xz cutting
plane





3D DES Simulation (Medium Grid) ...

Iso-Surface plot
of w-component
of velocity





3D DES Results

Grid Size	C_d	St
121x81x21	-----	-----
241x161x41	1.21	0.216
321x281x81	To be computed	
Exp.	0.99 ± 0.05	0.215 ± 0.005



Advantages of DES

- It eliminates the need for almost DNS type grid resolution near wall for LES simulations
- It provides computationally efficient LES computations



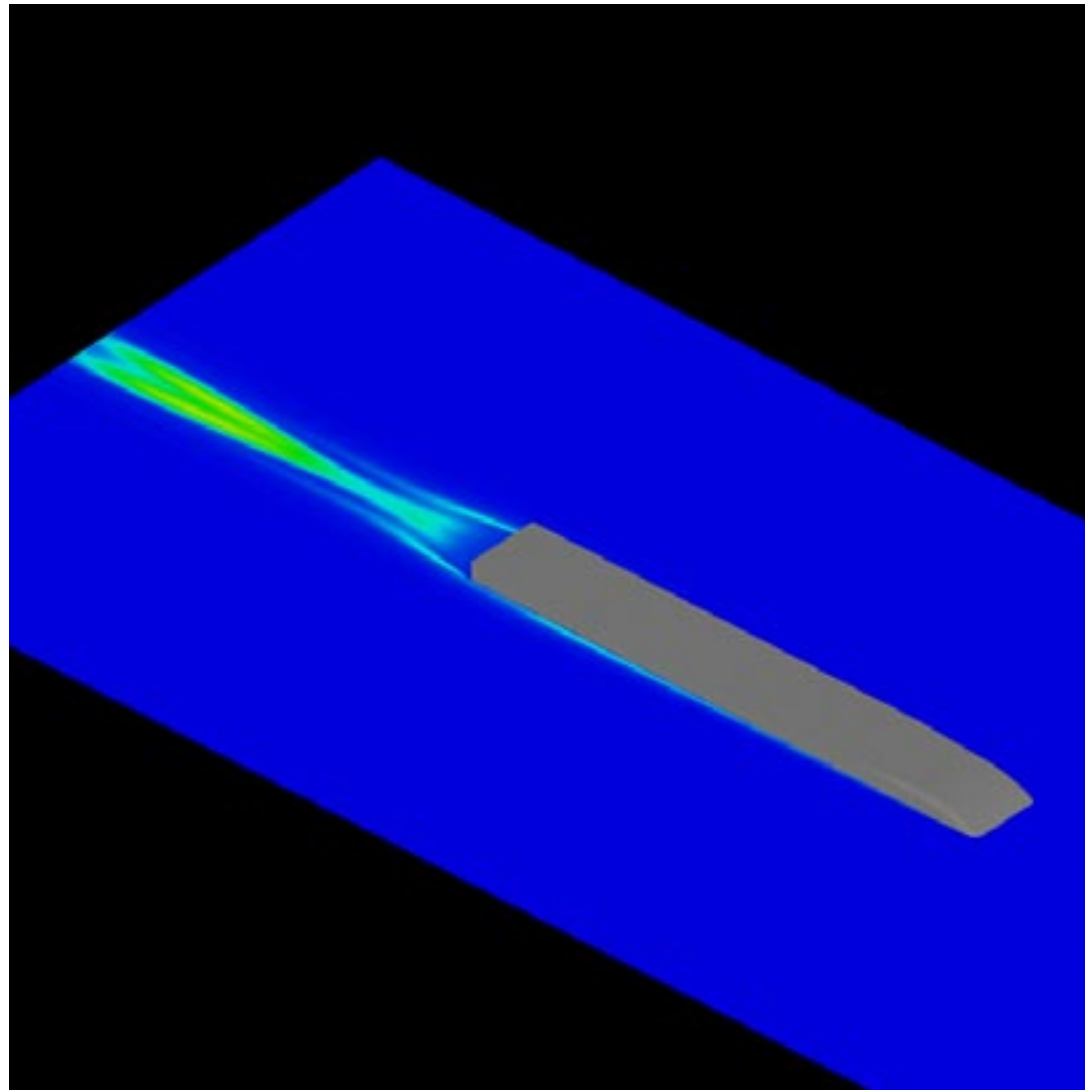
Disadvantages of DES

- The time-dependent wall boundary condition for LES is replaced with an unsteady RANS
- The accuracy of the RANS prediction is dependent on the type of RANS turbulence model used



GTS Flow Simulation, DES Result

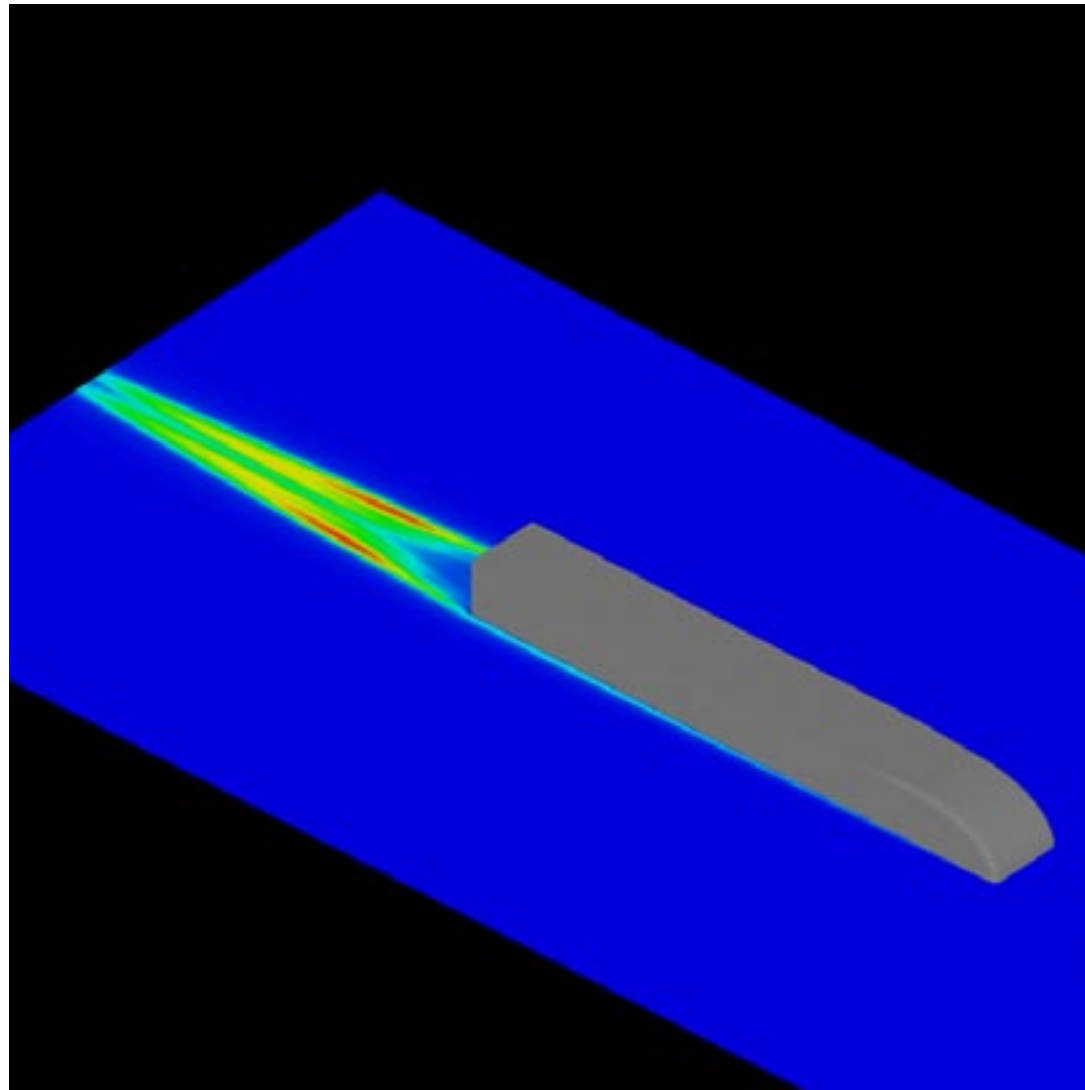
Contours of
turbulent viscosity,
xz cutting plane





GTS Flow Simulation, DES Result

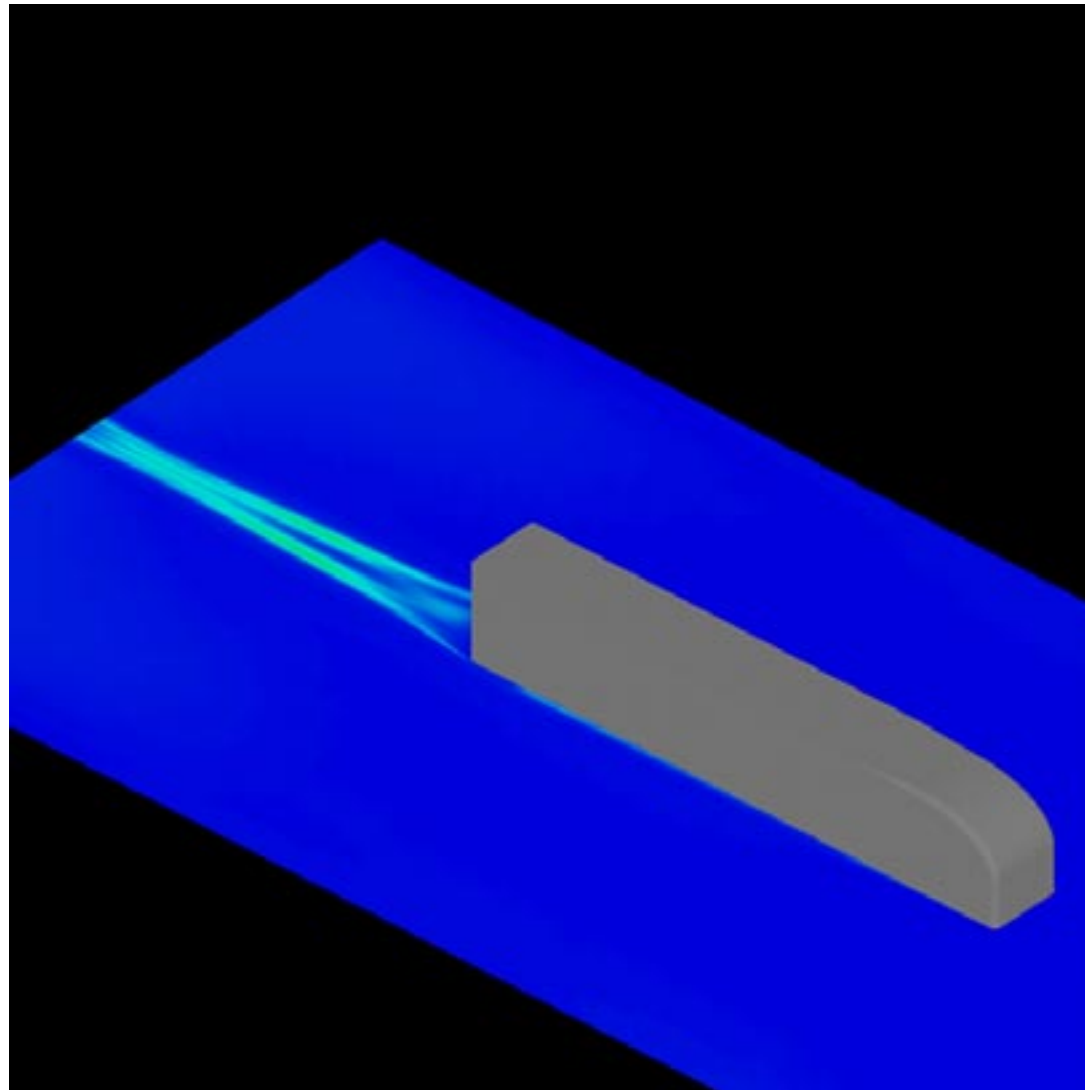
Contours of
turbulent viscosity,
xz cutting plane





GTS Flow Simulation, DES Result

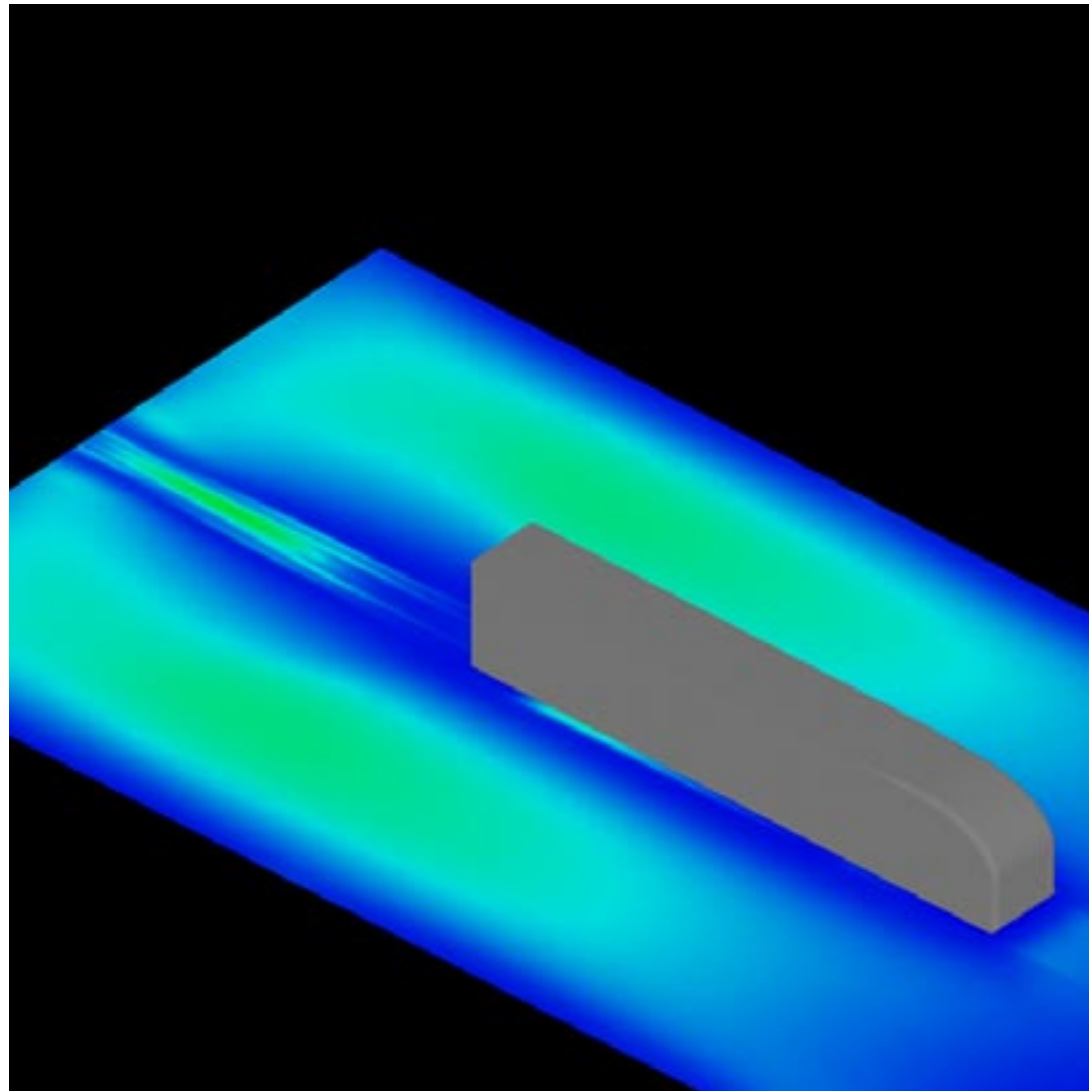
Contours of
turbulent viscosity,
xz cutting plane





GTS Flow Simulation, DES Result

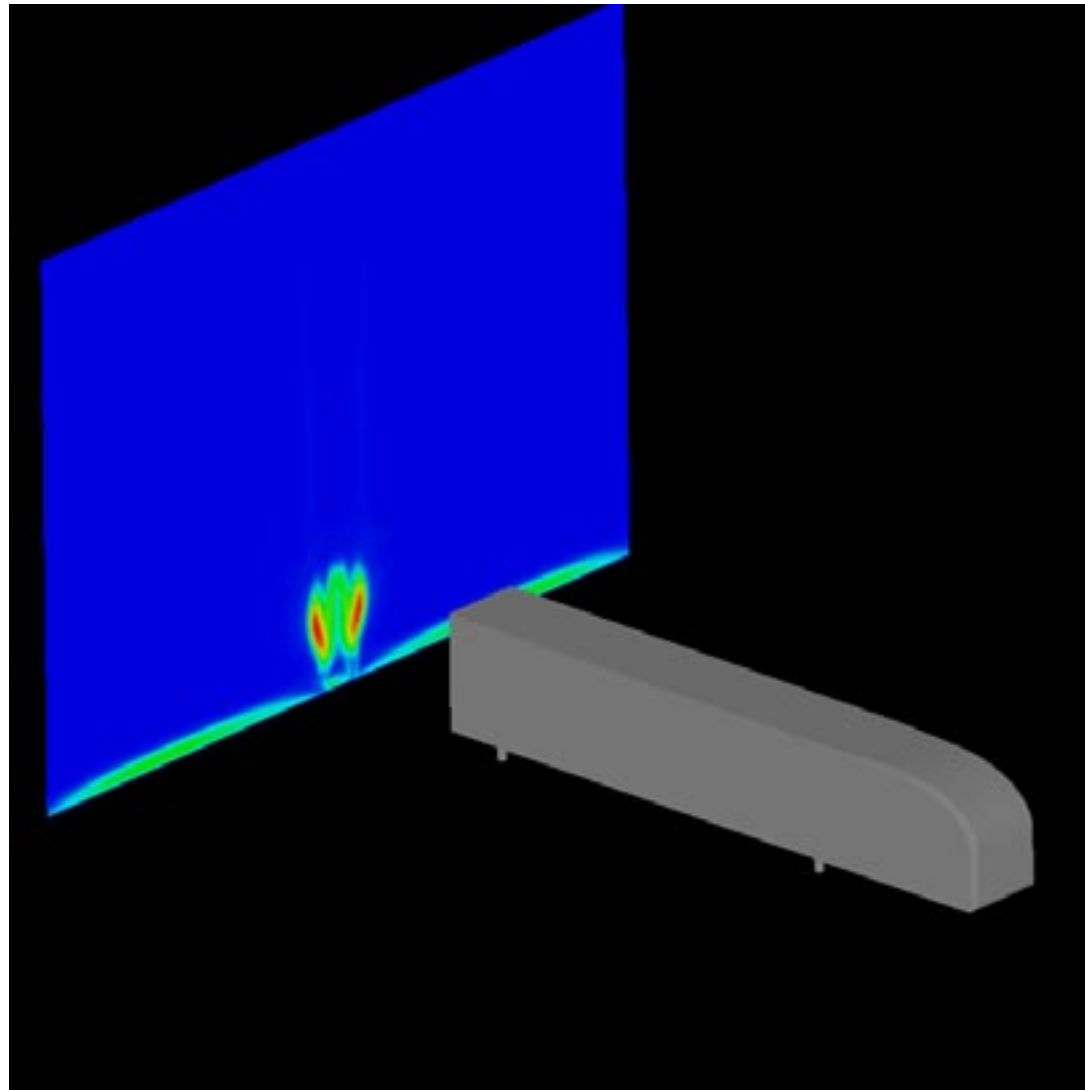
Contours of
turbulent viscosity,
xz cutting plane





GTS Flow Simulation, DES Result

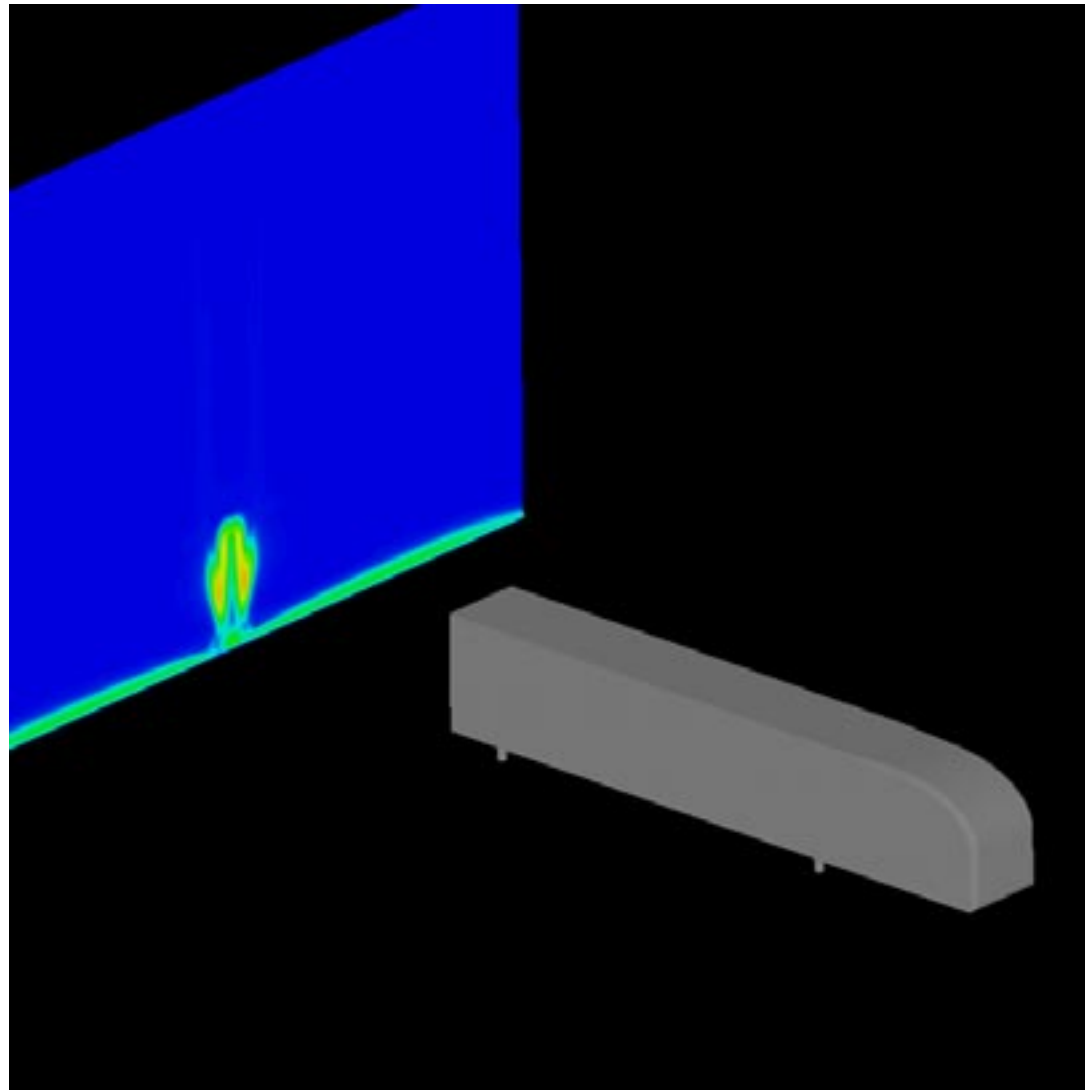
Contours of
turbulent viscosity,
yz cutting plane





GTS Flow Simulation, DES Result

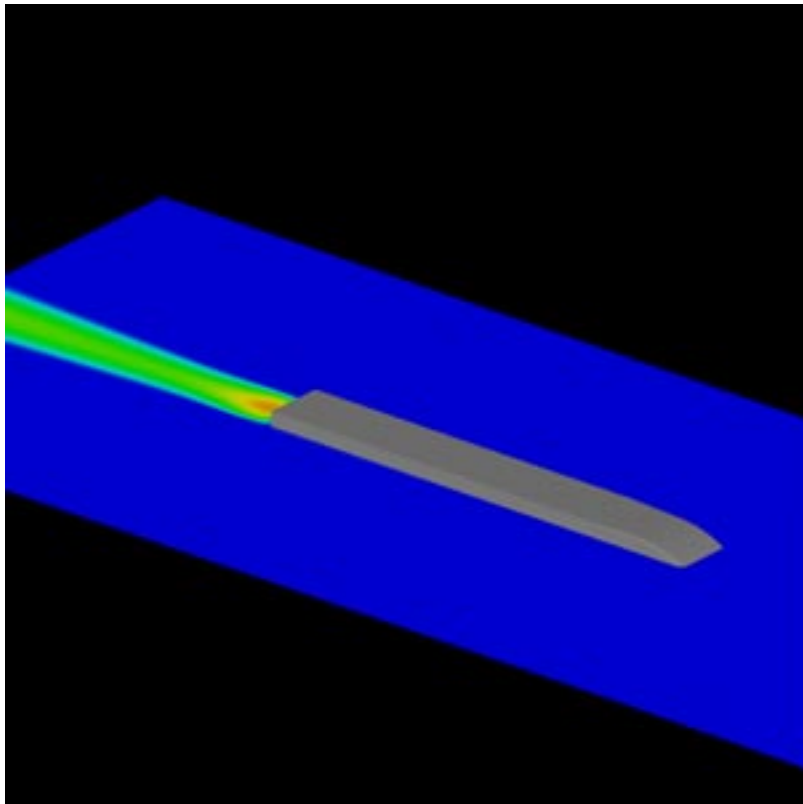
Contours of
turbulent viscosity,
yz cutting plane



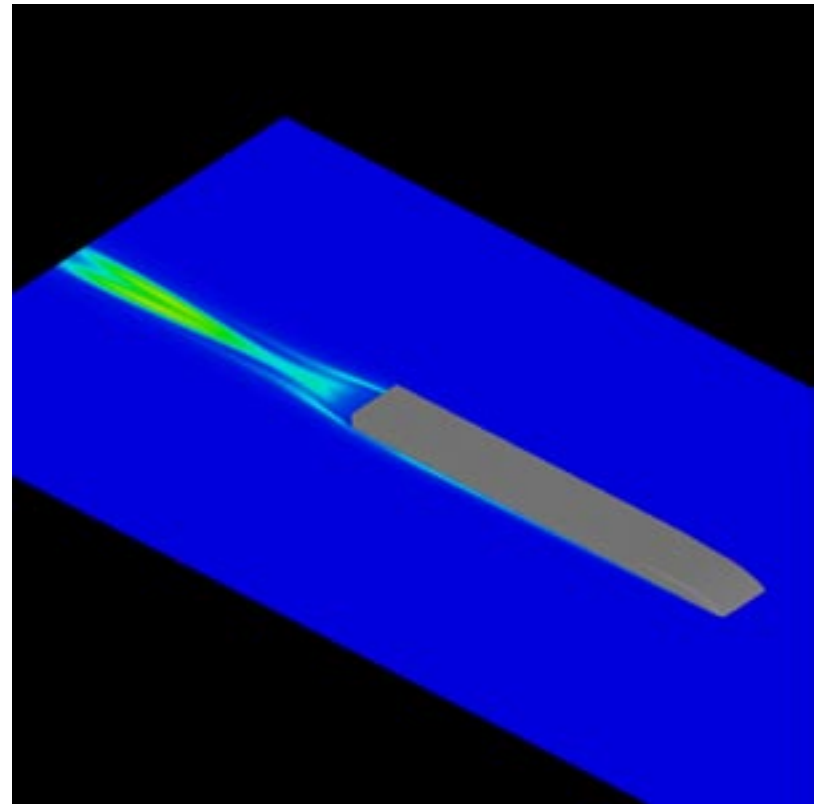


RANS vs. DES Simulations (GTS)

Steady RANS



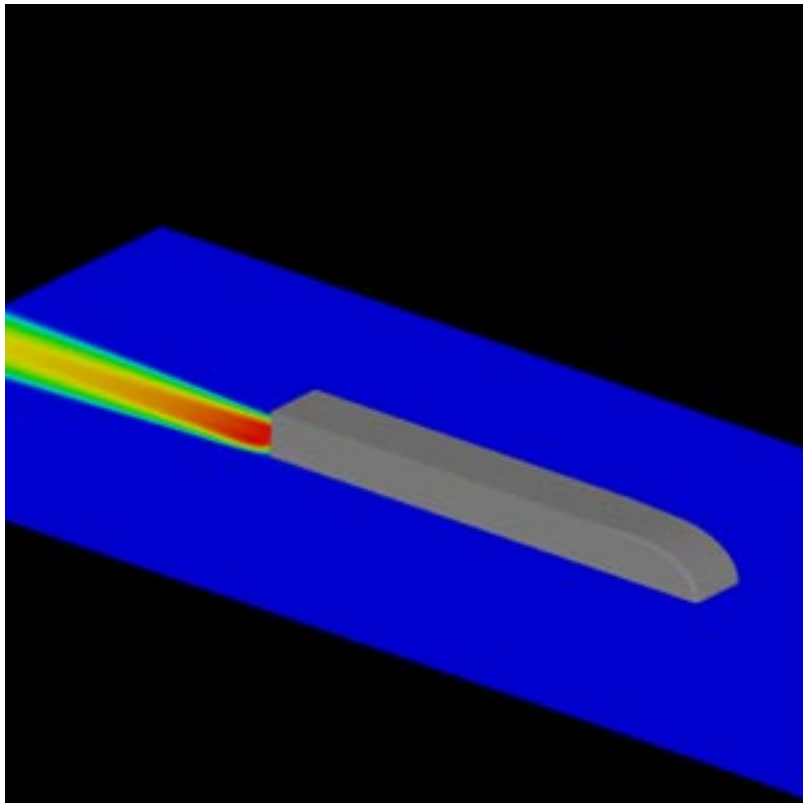
DES



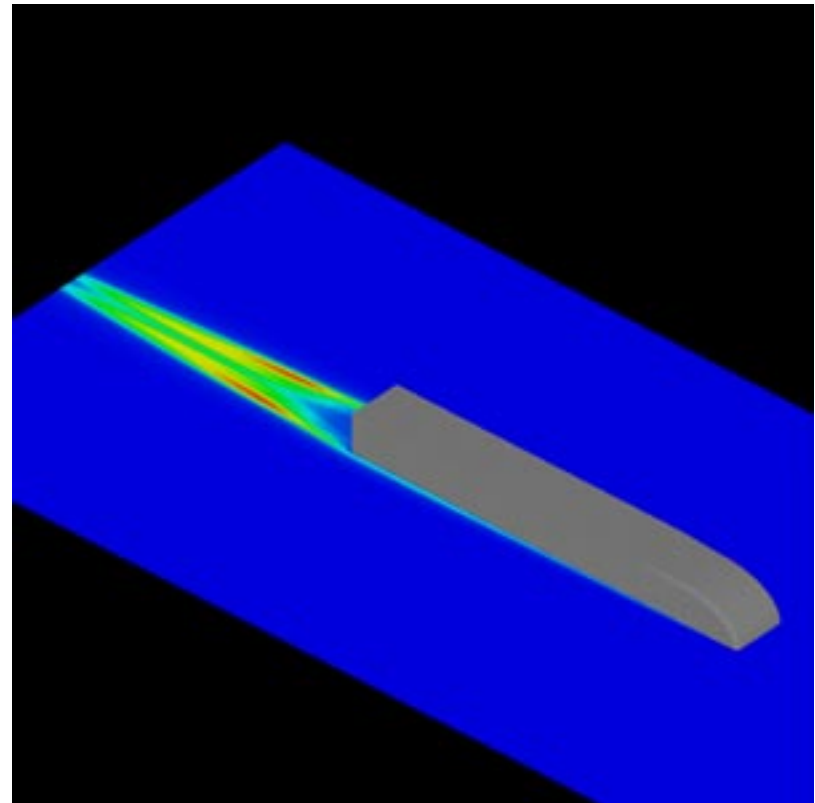


RANS vs. DES Simulations (GTS)

Steady RANS



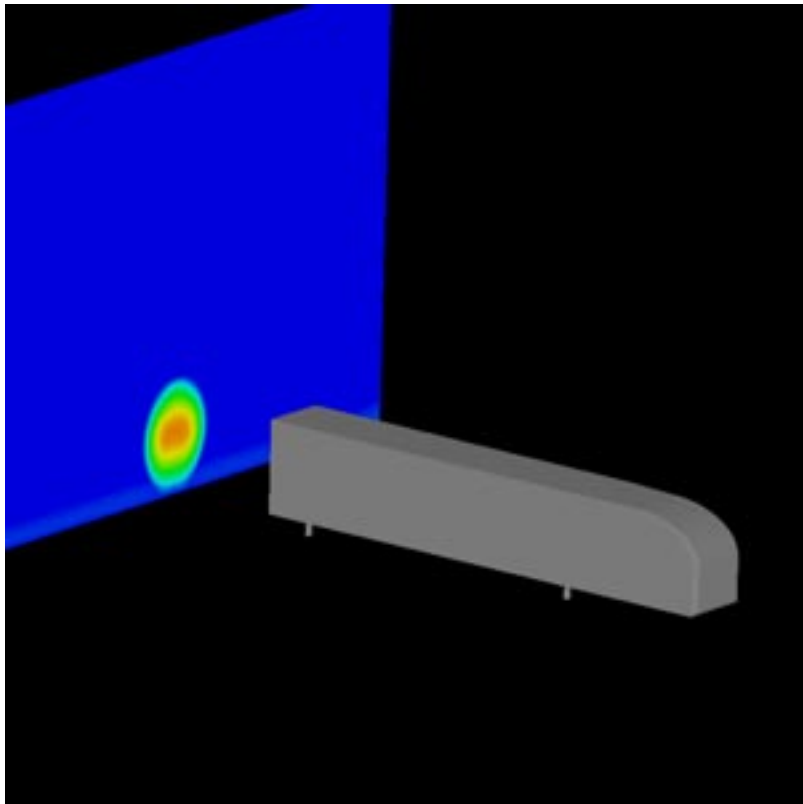
DES



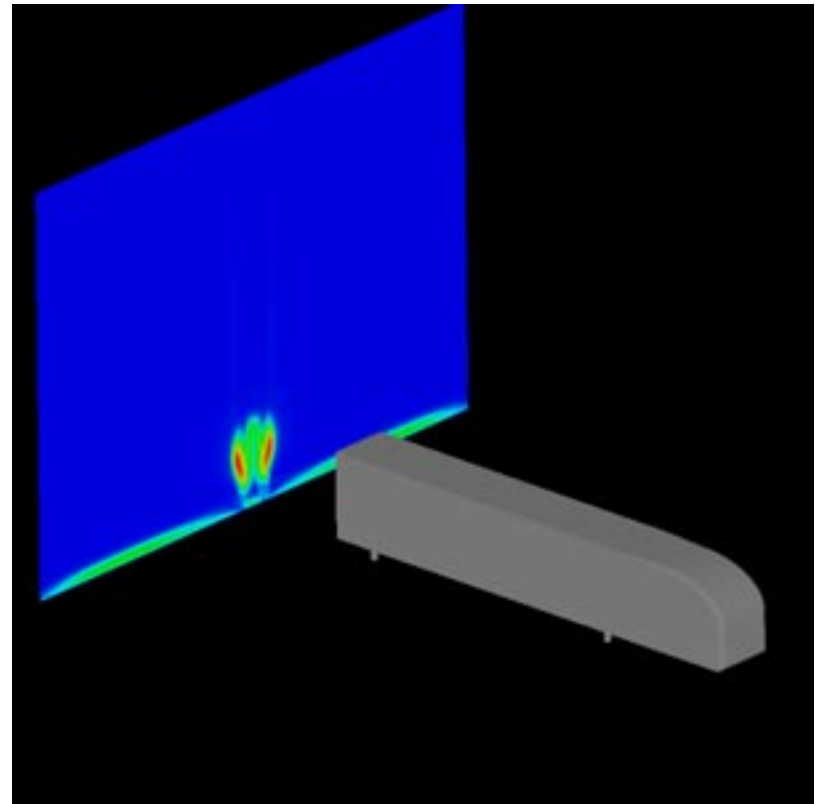


RANS vs. DES Simulations (GTS)

Steady RANS



DES





Conclusions

- **RANS produced very good results for the entire truck except for the base region**
- **Need to evaluate other eddy viscosity models for base flow using RANS**
- **DES approach may provide the best engineering prediction for the base flow**
- **Documentation**
 - **Provides opportunity for industry feedback**



Concluding Remarks

- **Grid resolution studies are still extremely important!**
- **Time resolution studies are also important (but not as widely practiced)**
- **Time averaging (or ensemble averaging) requires additional consideration**
- **For DES/LES calculations, solution is dependent on mesh size**



Sandia FY01 Tasks

1. Complete the SAE paper (March 2001) that documents the RANS results for 0°&10° yaws	
2. Perform grid resolution study for RANS calculations (NASA test)	
3. Construct appropriate grids for DES calculations of NASA test	
4. Perform DES calculations for the NASA 7x10 test at 0° yaw angles	\$135K
5. Document the DES and LES results in a SAE paper (March 2002) for the NASA 7x10 test at 0° yaw	
6. Investigate the predictive capability of RANS turbulent models ($k-\omega$, $k-\zeta$, SA) to model the wake flow of the GTS model in the NASA 7x10 test	
7. Help design and participate in future NASA wind tunnel experiments	\$330K

Incompressible Flow Modeling in ALE3D

Tim Dunn

New Technologies Engineering Division
Computational Physics Support Group
Lawrence Livermore National Laboratory



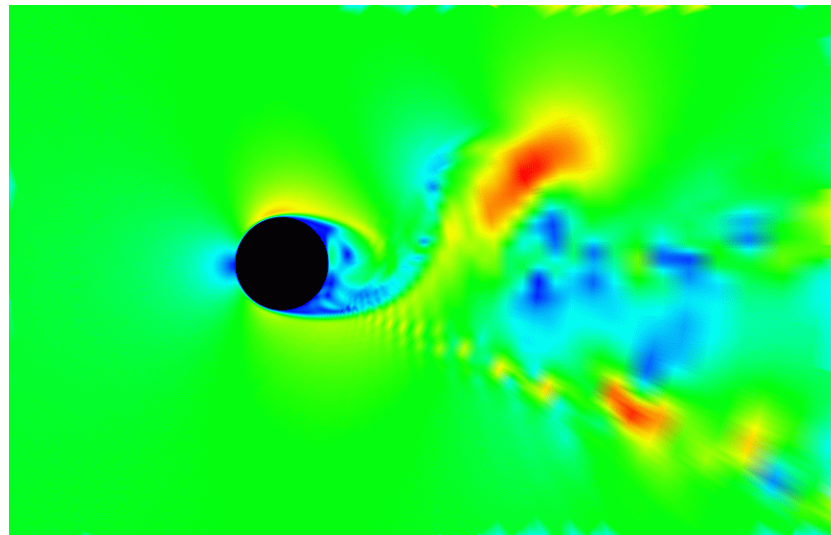
16-17 August 2000

DOE Truck Aero Team Meeting

Incompressible flow is currently running for small problems.



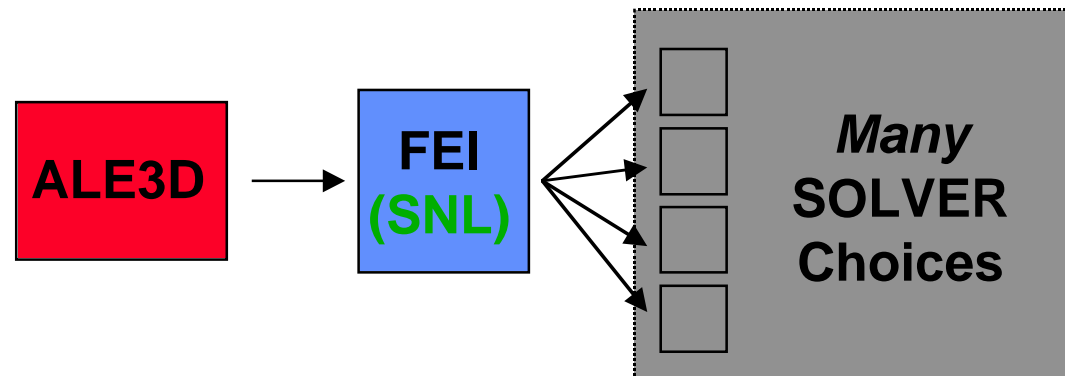
- Maximum problem size is currently limited to about 100,000 elements. Capability is increasing every day.
- Validation is underway.
- Working to decrease the runtime.



Most of the compute time is spent solving the matrix.



- The SNL Finite Element Interface (FEI) assembles and solves the system of equations.



- Some improvements need to be made:
 - Reduce redundant operations.
 - Add matrix stabilization to help the convergence rate of iterative solvers.
 - Work with LLNL solver experts to develop better preconditioning and advanced solver methods.

Some additional capabilities are being considered.



- **Implicit Time-Integration**
 - **Explicit scheme requires very small timesteps**
 - Semi-implicit (Diffusion terms implicit)
 - Full-implicit
- **Turbulence Modeling**
 - **Wall models**
 - No damping (current approach)
 - Near-wall damping
 - DES

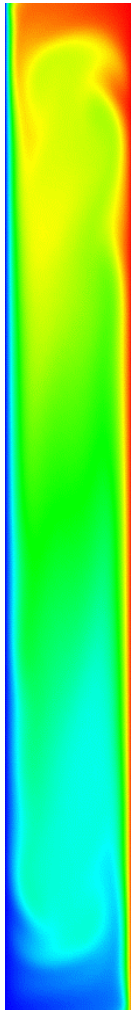
Fluid/Solid Heat Transfer is now available.



- Incompressible flow is coupled with thermal
- Loosely coupled approach (flow and temperature solved separately)
- Boussinesq approximation for Buoyancy:

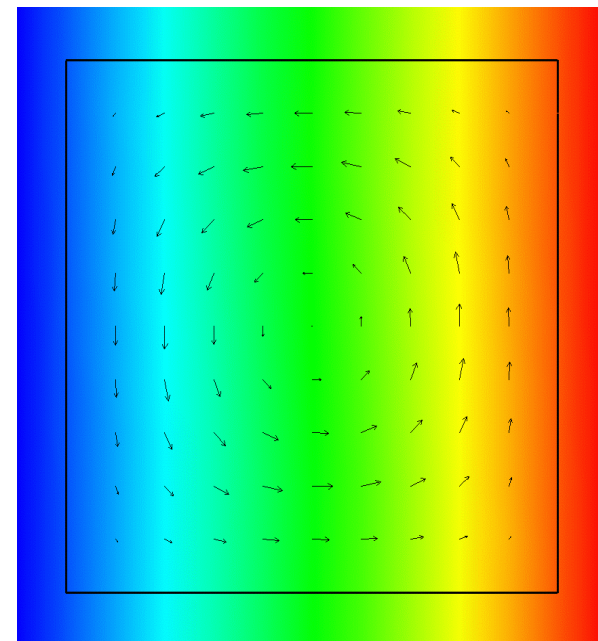
$$g(1 - \beta(T - T_0))$$

Critical Ra No.
flow



Thermally-Driven Cavity
Inside Solid Box

COLD WALL



HOT WALL

Particle Tracking Algorithm is now available.



- Dilute Particle Flow
 - Particle motion is controlled by local aerodynamic forces
 - No particle-particle interactions
 - Particles do not influence the carrier fluid
- Trajectory is tracked by solving the Lagrangian equation of particle motion

$$\ddot{x}_i = -\frac{3}{4} \frac{\rho_f}{\rho_p} \frac{C_D}{d} |\tilde{\mathbf{v}} - \tilde{\mathbf{u}}| (v_i - u_i) + \tilde{g} + \dots$$

Gravitational Force (red text with arrow pointing to \tilde{g})
Aerodynamic Drag (blue text with arrow pointing to $\frac{3}{4} \frac{\rho_f}{\rho_p} \frac{C_D}{d}$)

- The second order ODE is solved with standard numerical techniques
- Additional terms can be added to the right hand side to account for more difficult particle physics or turbulent fluctuations in the carrier fluid

The Incompressible Flow uses an Eulerian Formulation



- Galerkin Finite-Element Method
 - 8-node Hexahedral Elements
 - Tri-linear Basis Functions for Velocity
 - Piecewise Constant Pressure
 - Single Point Integration

The Incompressible Flow Equations.



- Conservative Form

Mass	$\frac{\partial u_\alpha}{\partial x_\alpha} = 0$	$\tau_{\alpha\beta} = -P\delta_{\alpha\beta} + \nu \frac{\partial u_\alpha}{\partial x_\beta}$
Momentum	$\frac{\partial u_\alpha}{\partial t} + u_\beta \frac{\partial u_\alpha}{\partial x_\beta} = \frac{\partial \tau_{\alpha\beta}}{\partial x_\beta}$	$P = \frac{p}{\rho}$

- Variational Form

Mass	$\int_{\Omega} w \frac{\partial u_\alpha}{\partial x_\alpha} = 0$
Momentum	$\int_{\Omega} v \left(\frac{\partial u_\alpha}{\partial t} + u_\beta \frac{\partial u_\alpha}{\partial x_\beta} \right) + \int_{\Omega} \tau_{\alpha\beta} \frac{\partial v}{\partial x_\beta} = \int_{\partial\Omega} v f_\alpha$
	$f_\alpha = n_\beta \tau_{\alpha\beta}$

The Incompressible Flow Equations: Discrete Form.



- Variable Expansion

$$u_{\alpha}^h(x, t) = \sum_{j=1}^N u_{\alpha}^j(t) \varphi_j(x)$$

$$v = \varphi_i(x), \quad i = 1, N$$

$$P^h(x, t) = \sum_{j=1}^M P_j(t) \varphi_j(x)$$

$$w = \psi_i(x), \quad i = 1, M$$

Mass $\left(\int_{\Omega} \psi_i \frac{\partial \varphi_j}{\partial x_{\alpha}} \right) u_{\alpha}^j = 0$

Momentum $\left(\int_{\Omega} \varphi_i \varphi_j \right) \frac{\partial u_{\alpha}^j}{\partial t} + \left(u_{\beta}^k \int_{\Omega} \varphi_i \varphi_k \frac{\partial \varphi_j}{\partial x_{\beta}} \right) u_{\alpha}^j + \left(\int_{\Omega} v \frac{\partial \varphi_i}{\partial x_{\beta}} \frac{\partial \varphi_j}{\partial x_{\beta}} \right) u_{\alpha}^j - \left(\int_{\Omega} \psi_j \frac{\partial \varphi_i}{\partial x_{\alpha}} \right) P^j = \int_{\partial \Omega} \varphi_i f_{\alpha}$

The Incompressible Flow Equations: Matrix Form.



$$C^T \underline{v} = 0$$

$$M\dot{u} + [K + N(u)]u + CP = F$$

OR

$$\begin{bmatrix} M' & C \\ C^T & 0 \end{bmatrix} \begin{bmatrix} v \\ P \end{bmatrix} = \begin{bmatrix} F' \\ 0 \end{bmatrix}, \quad [C^T M'^{-1} C][P] = C^T M'^{-1} F$$

where

$$\underline{v} = \underline{u}^{n+1} - \underline{u}^n \quad M' = \frac{M}{\Delta t} \quad F' = F - [K + N(u)]u$$

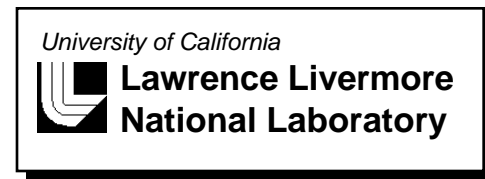
$$C = \begin{bmatrix} c_{in(1)} \\ c_{in(2)} \\ c_{in(3)} \end{bmatrix}; c_{in(\alpha)} = \int_{\Omega} \psi_n \frac{\partial \phi_i}{\partial x_{\alpha}} \quad M = \begin{bmatrix} m_{ij} & 0 & 0 \\ 0 & m_{ij} & 0 \\ 0 & 0 & m_{ij} \end{bmatrix}; m_{ij} = \int_{\Omega} \phi_i \phi_j$$

Progress in LES for Heavy Vehicle Aerodynamics for the August 2000 Aerodrag Group Meeting

Dan Flowers, Rose McCallen, Tim Dunn, Jerry Owens, Greg Laskowski

**Lawrence Livermore National Laboratory
Livermore, CA**

August 2000



Introduction

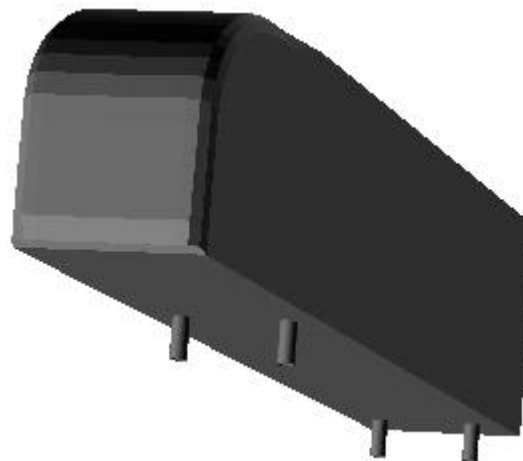


Goal

To develop simulation tools which can accurately predict the flow-field of heavy vehicles. These tools can be applied to investigating drag reduction strategies.

Approach

Carefully validate the simulations with well characterized experimental data



Several approaches are being used to simulate the GTS



SNL

Reynolds Average Navier-Stokes (RANS)/ **Detached Eddy Simulation (DES)**
Compressible Finite Volume Code
Average “Steady” Solution/**Unsteady Solution**
Widely used - may not predict drag correctly

LLNL

Large Eddy Simulation (LES)
Incompressible\Compressible Finite Element Code
Unsteady Solution of large scales/approximation of the small scales
Computationally intensive

Caltech

Direct Numerical Simulation/ **LES**
Vortex Method
Gridless
In development

Large Eddy Simulation (LES) is an advanced method of simulating turbulent flow



- Direct Numerical Simulation of turbulent flow for useful Reynolds numbers is beyond current computational capabilities
 - Wide range of scales
 - Long runtimes
- LES is a compromise between resolution and available computational resources
 - Large scales are resolved and solved directly
 - Small, dissipative, scales are modeled using a 1-parameter “sub-grid-scale” (SGS) model

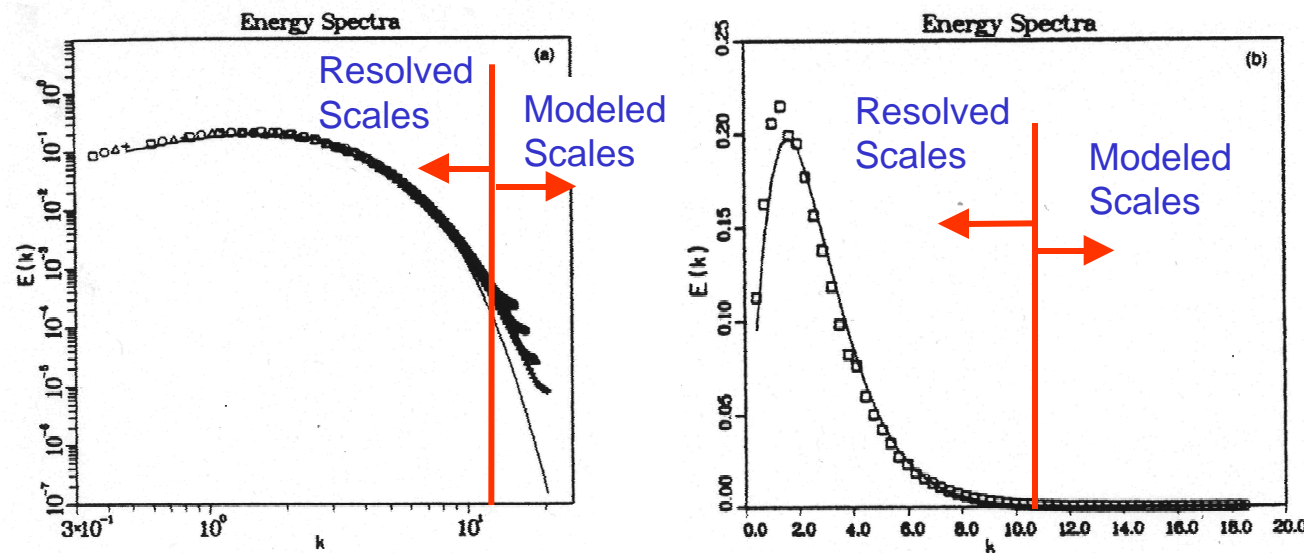


Fig.3 Energy spectrum (logarithmic scale left and linear scale right graph) according to Domaradski and Rogallo (1990).

We are focusing on application of our code as a trustworthy simulation tool

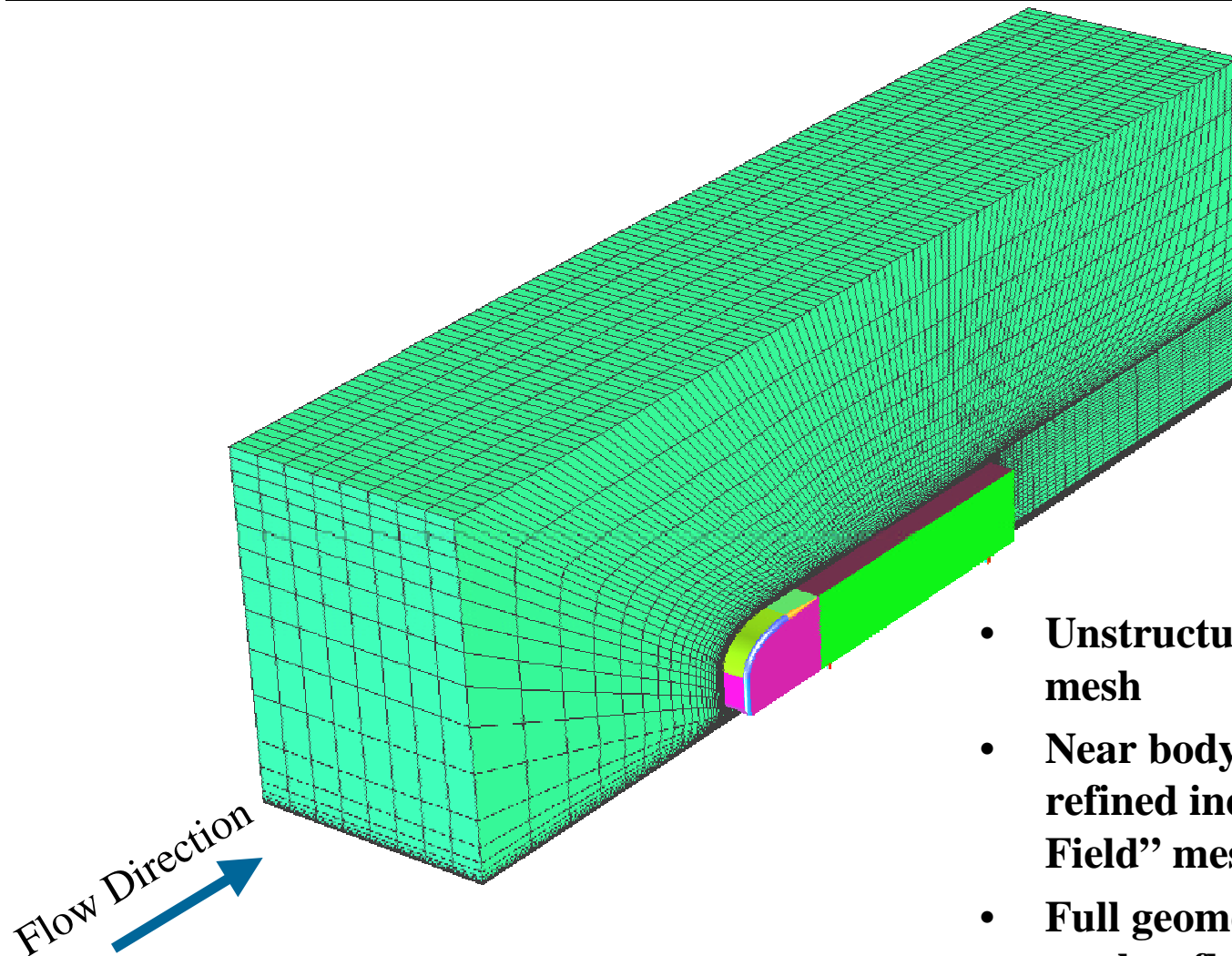


Compressible/Incompressible simulations of 7' x 10' Data

Effect of Grid on Corner Separation

Incompressible “Tunnel” Empty Simulations

Finite element mesh

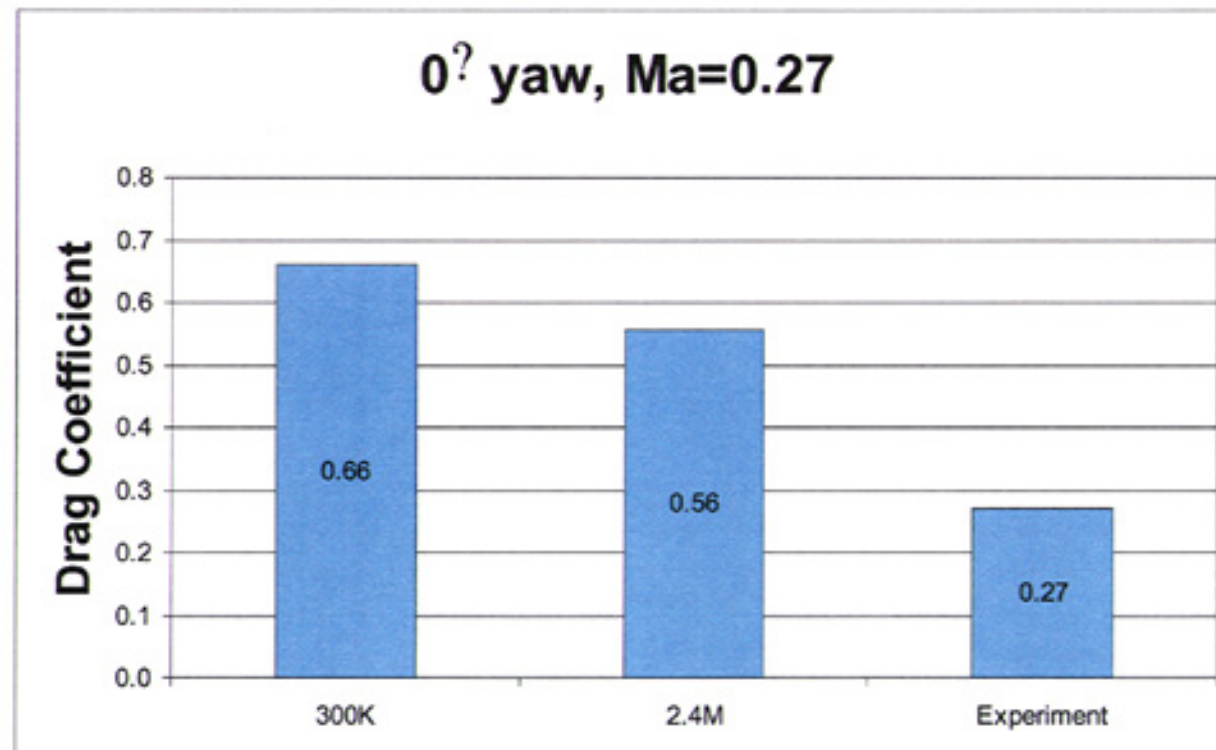


- **Unstructured multi-block mesh**
- **Near body mesh can be refined independently of “Far-Field” mesh**
- **Full geometry simulated - mesh reflected about symmetry plane**

Compressible flow simulations with for 0° yaw have been performed with 2 grids



- 300K and 2.4 Million Elements
- Drag coefficient is overpredicted compared to experiment
- We think we know the culprit in this discrepancy



Simulation results - time average flow field

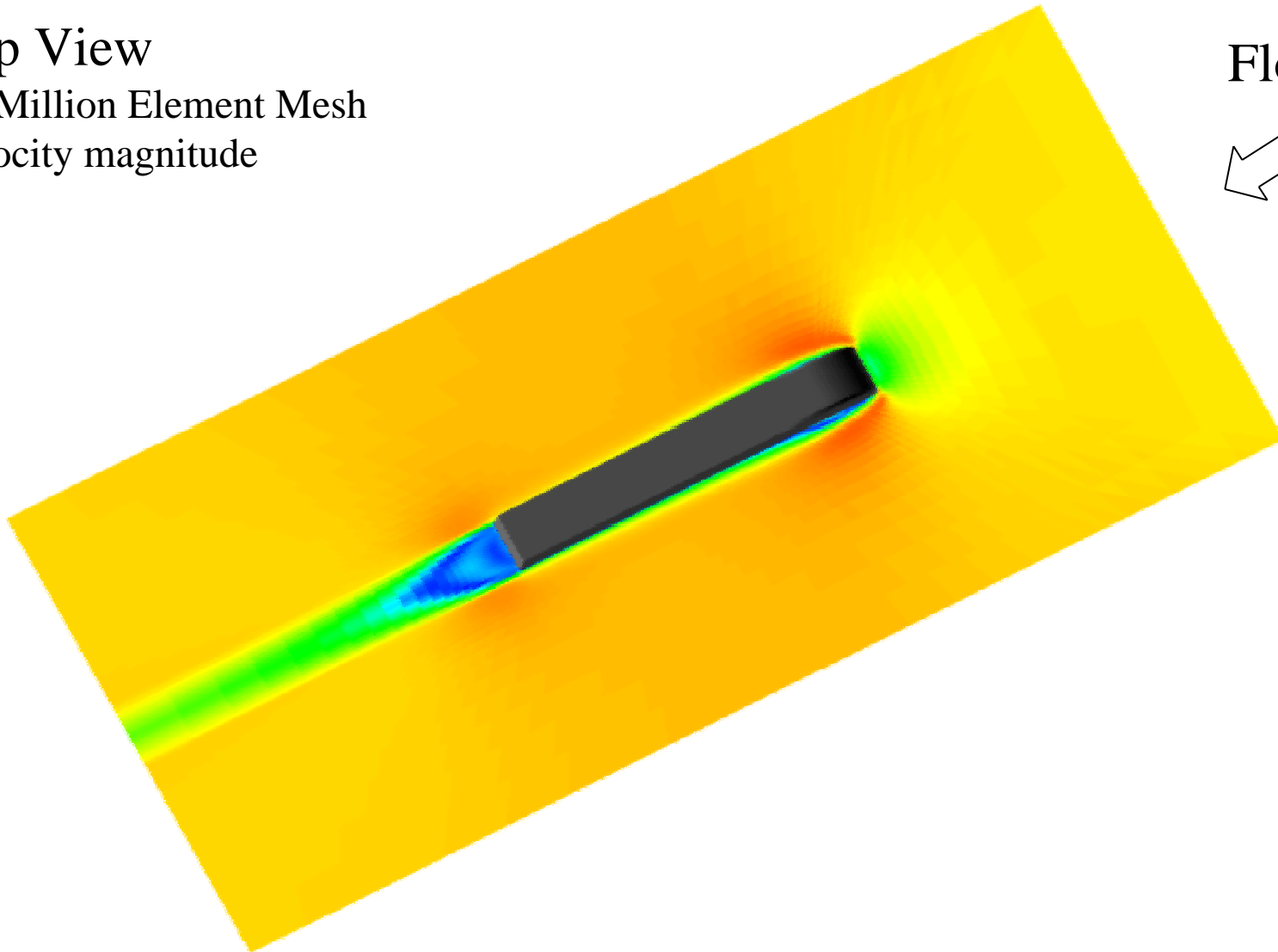
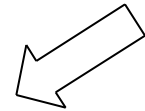


Top View

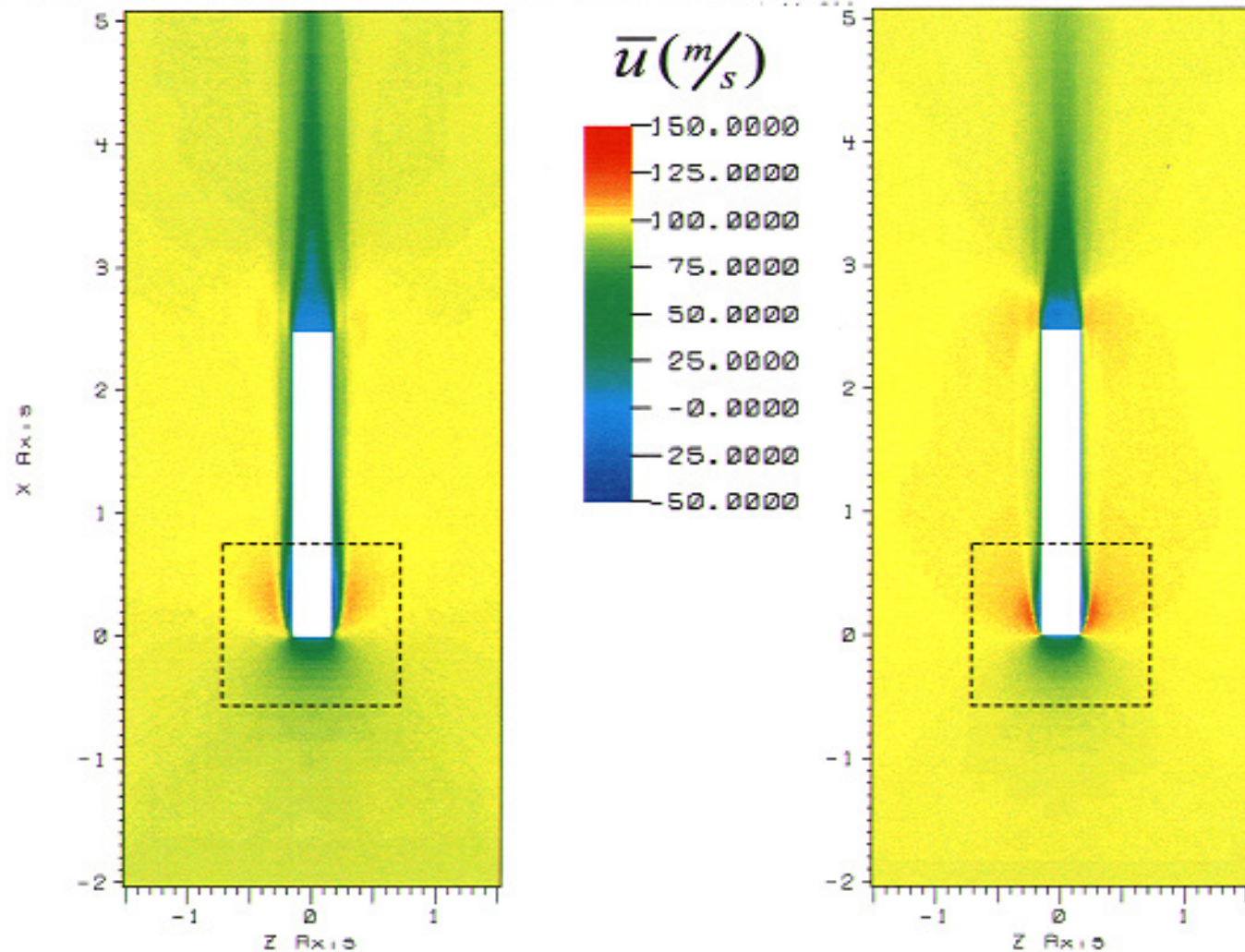
0.3 Million Element Mesh

Velocity magnitude

Flow



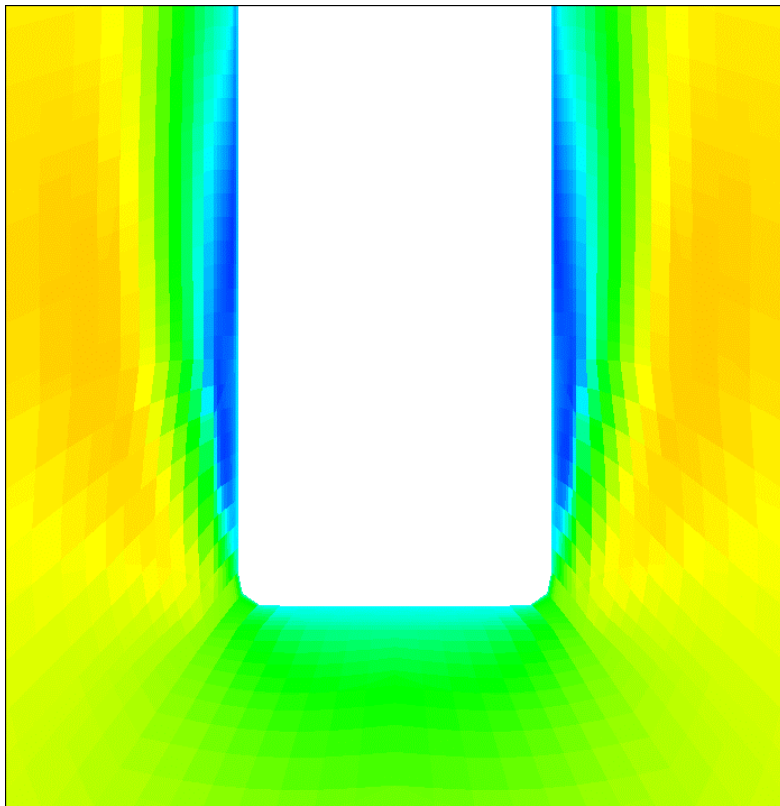
Time average results show some differences between the course and less course grid



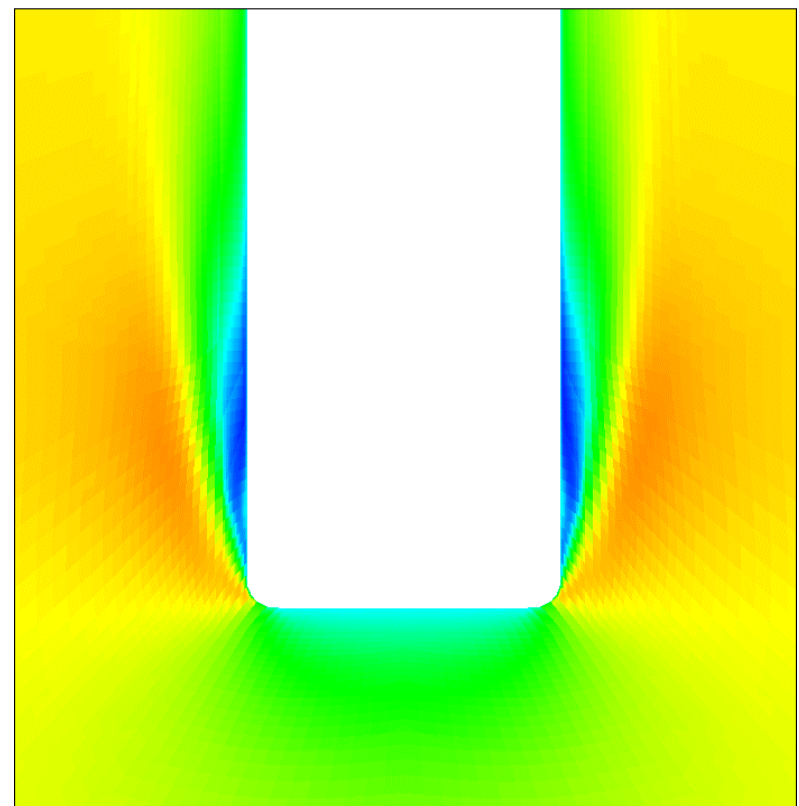
Course Mesh

Less Course Mesh

The recirculation zones decrease in size as the grid is refined



Course Mesh



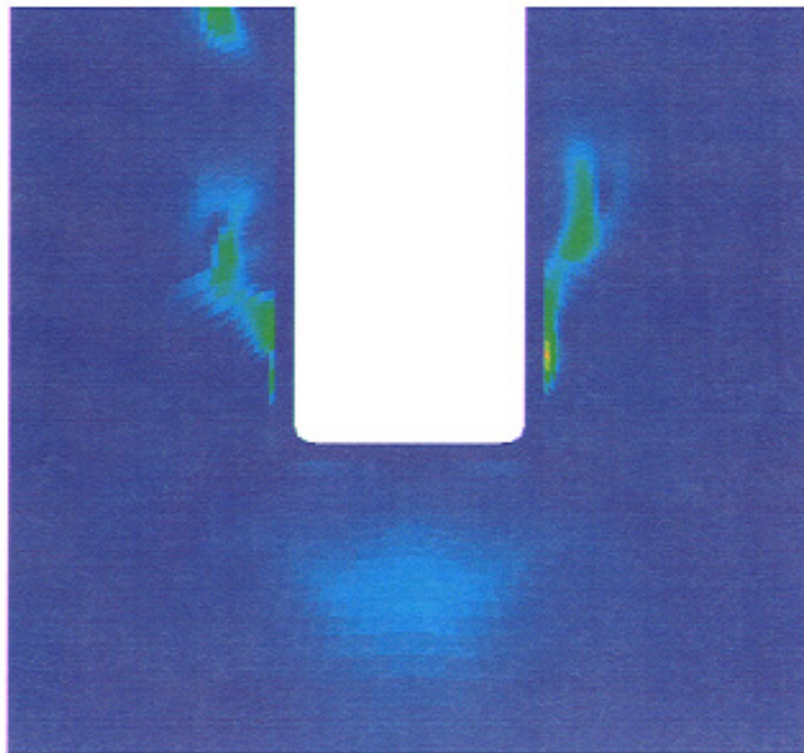
Less Course Mesh

The effect of the length-scale choice for the Smagorinsky model are being investigated

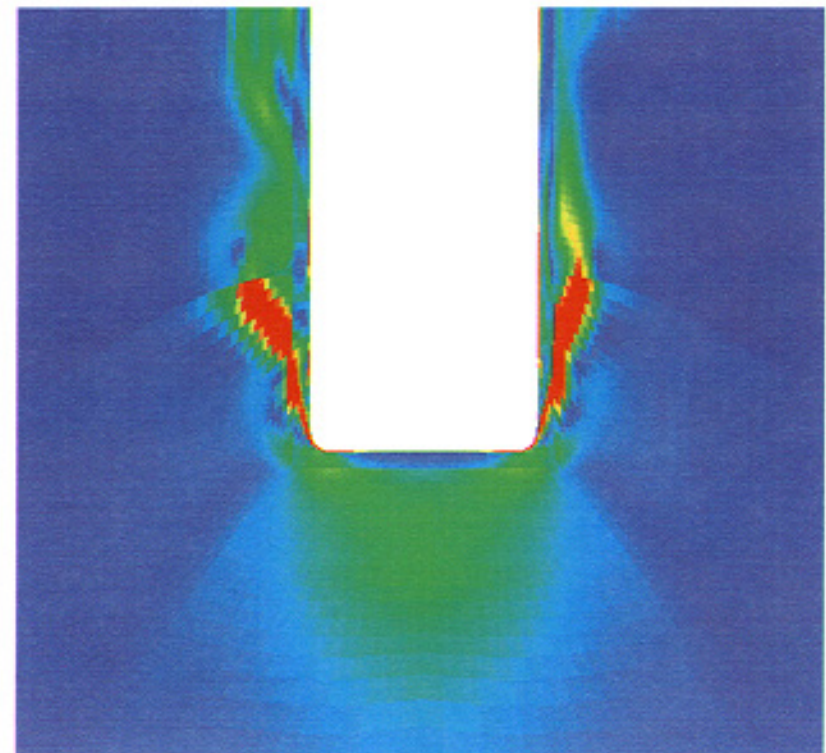


- Smagorinsky Model

$$\nu_t = (c_s \Delta)^2 \sqrt{S_{\alpha\beta} S_{\alpha\beta}} \cdot f(d)$$

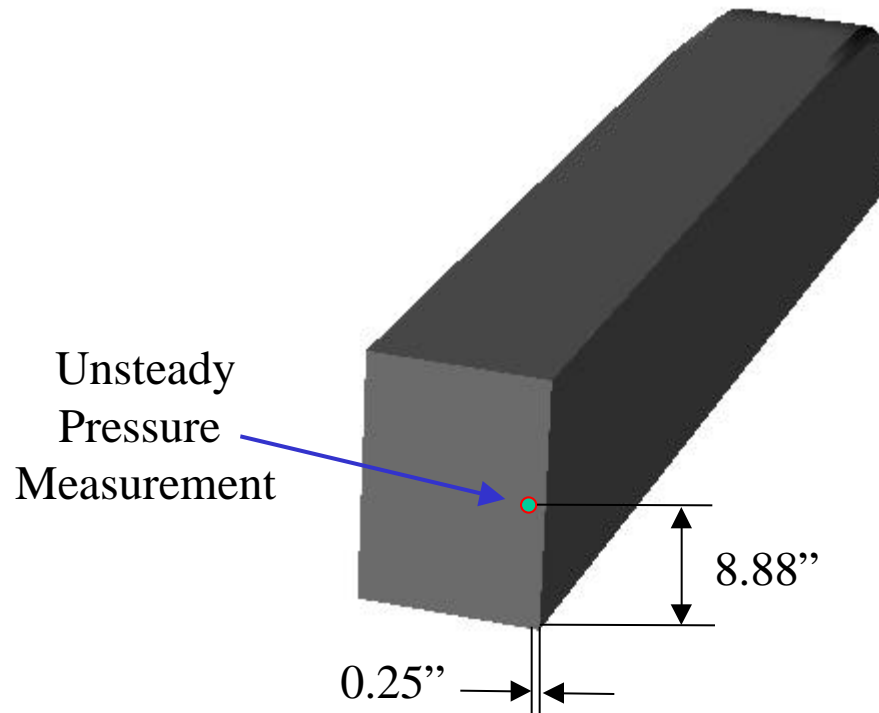


$\Delta = \text{min element distance}$



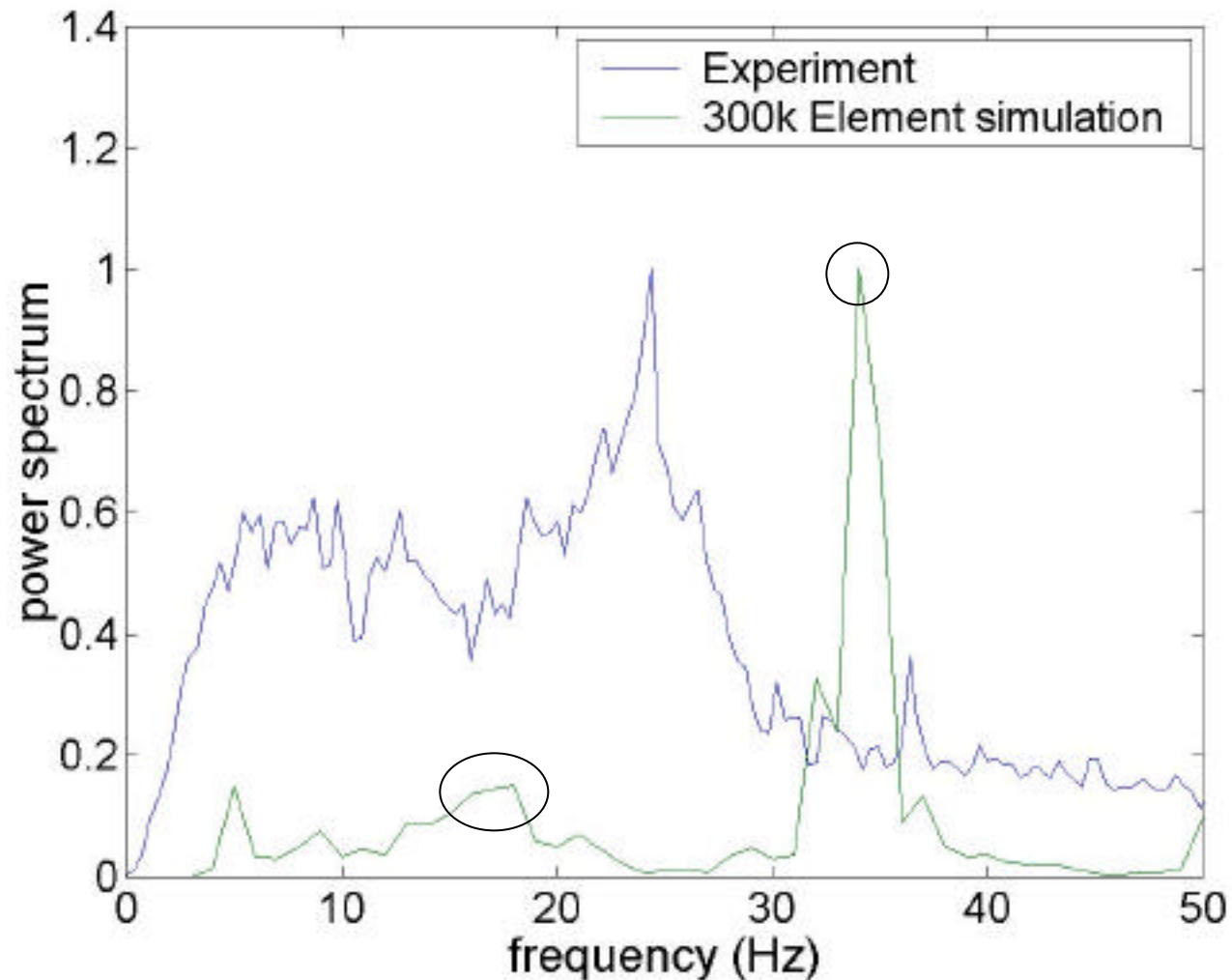
$\Delta = \text{cube root of element vol}$

Unsteady pressure is recorded in simulation for comparison to experimental data

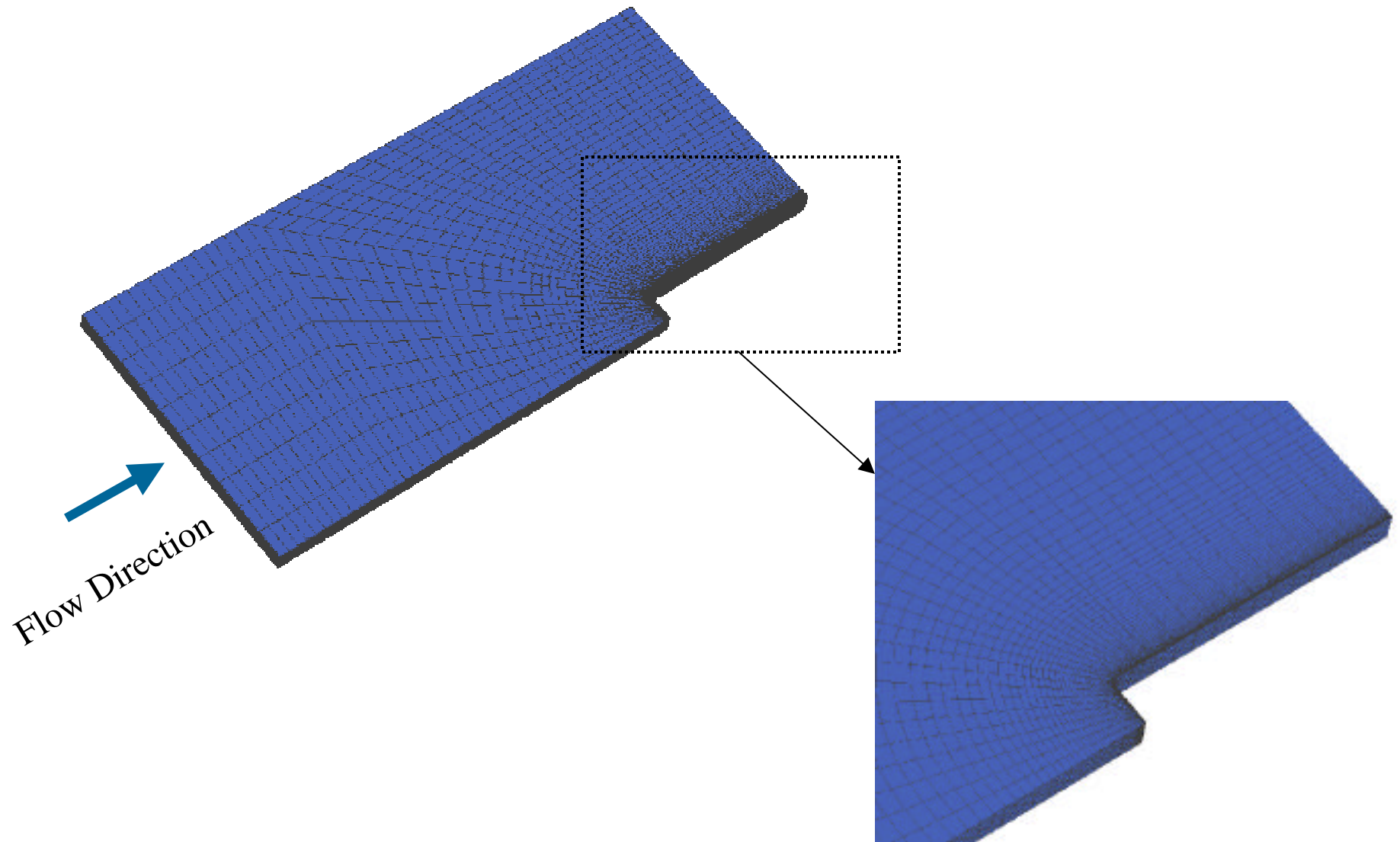


- **Unsteady pressure transducer location used in experiments and simulations**

The dominant frequency component from simulations is shifted from the dominant peak in the experiments



We are now investigating the effect of the grid resolution on the front corner



Tunnel empty simulations are necessary for careful matching of experimental conditions

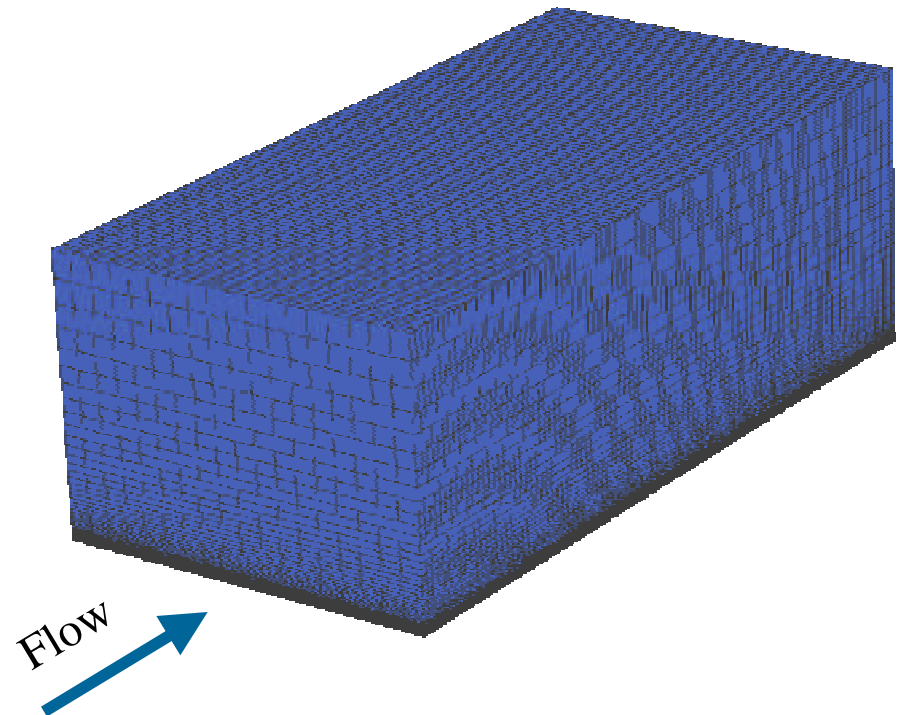


Simulations being conducted with sister code

- Incompressible
- Finite Element
- **Implicit**
- **Structured**
- Smagorinsky SGS
- van Driest model for wall

Tunnel is treated as a box

- Tunnel Q is based on experiment
- Top and side walls have slip BC
- No slip on floor
- Specified velocity profile inflow
- Zero natural boundary condition for outflow



We are making a great deal of progress in our application of LES to the truck flow problem



Compressible simulations of 7' x 10' Data

- Have been completed for two grids at 0° yaw
- Corner improvements, corrected Reynolds number, and improved eddy viscosity for future runs

Effect of Grid on Corner Separation

Are being conducted to determine resolution needed to correctly capture the front corner flow

Incompressible “Tunnel” Empty Simulations

Are providing information on appropriate boundary conditions as well as insight into the application of wall models

Movie in the Morning System

An Overview

Jerry Owens

LLNL

The Problem and Some Solutions

- It is hard to understand what what is going on in our large production problems
- Possible solutions
 - Interactively browse lots of very large files
 - Make movies during the production and let users view the movie in the morning

Goals for the Movie in the Morning System (MIMS)

- Provide users with the information they need as quickly as possible
- Allow users to see how the problem evolves over time
- Produce movies while the code runs
- Produce several movies per problem each night

MIMS Goals (Cont.)

- Deliver movies to the user via web
- Produce high resolution versions of each frame in the movie as the movie is being produced
- Allow easy access to any of the high resolution frames via the web

MIMS Goals (Cont.)

- **Manage all the details of producing the movies without user interaction**
- **Produce the movies at the times requested**
- **Be smart about production job resubmission**
- **Make each movie as separate batch job**
- **Keep a log of everything it does**

How MIMS Works

- User adds two lines to the batch job request
- First line adds a process that checks if it is time to make a movie and if so generates the scripts for making the movie from template files and launches the batch jobs that produce the movie

How MIMS Works (Cont.)

- The second line starts a process checks for problems in the runs and then either resubmits the problem to continue the run or resubmits it to run up to the error time and stops resubmitting the problem

How MIMS Works (Cont.)

- The frames for the movies and the high resolution TIF files are produced on the same machine at the production is run
- MIMS has daemons on other machine that do the conversion to the movie format

MIMS Status

- It is currently producing up to six movies for two currently running production problems

Implementation of Spalart- Allmaras Model into ALE3D for Detached Eddy Simulation (DES)



Greg Laskowski

PhD Candidate
Stanford University

Summer Intern
Lawrence Livermore National Lab

August 16, 2000

SGS Stress Term

- Currently handled using Smagorinsky model
 - Difficulty near solid surfaces
 - Questionable for separated flows
 - Requires high degree of empiricism
 - Smagorinsky constant
 - Van Driest damping near solid surfaces
- Implement variant of Spalart-Allmaras model for DES
 - RANS near wall
 - LES away from wall

Spalart-Allmaras Model

$$\nu_{SGS} = \tilde{\nu} f_{v1}$$

$$\frac{D\tilde{\nu}}{Dt} = c_{b1} \bar{S} \tilde{\nu} - c_{w1} f_w \left(\frac{\tilde{\nu}}{d} \right)^2 + \frac{1}{\sigma} \frac{\partial}{\partial x_k} \left[\left(\nu + \tilde{\nu} \right) \frac{\partial \tilde{\nu}}{\partial x_k} \right] + \frac{c_{b2}}{\sigma} \frac{\partial \tilde{\nu}}{\partial x_k} \frac{\partial \tilde{\nu}}{\partial x_k}$$

$$f_{v1} = \frac{X^3}{X^3 + c_{v1}^3}; \quad f_{v2} = 1 - \frac{X}{1 + X f_{v1}}; \quad f_w = g \left[\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{1/6}$$

$$X = \frac{\tilde{\nu}}{\nu}; \quad g = r + c_{w2} (r^6 - r); \quad r = \frac{\tilde{\nu}}{\bar{S} \kappa^2 d^2}$$

$$\bar{S} = S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}; \quad S = \sqrt{2 \Omega_{ij} \Omega_{ij}}; \quad \Omega_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right)$$

Source and Sink Terms



- Highly non-linear
- Depend on “d”
 - **RANS** formulation: “d” is minimum distance to solid surface
 - **DES** formulation: “d”
 - $\min(d_{\text{wall}}, C_{\text{DES}}\Delta)$
 - Where $\Delta = 1/3V_{\text{element}}$ and $C_{\text{DES}} = 0.65$

Implementation Procedure

- Must implement in ALE3D (FEM code)
- Procedure:

- Step 1. SA in variational Form ✓
 - A) Explicit formulation
 - B) Implicit formulation
- Step 2. Implement explicit form ✓
- Step 3. Validate explicit form Ø
- Step 4. Implement implicit form #
- Step 5. Validate implicit form #

✓: Completed; Ø In progress; # To be done (if necessary)

Validation Effort

- Plane channel flow simulation
 - $Re_\tau = 2000$
 - $N_x \ N_y \ N_z = 64 \times 64 \times 32$
 - $\Delta x^+ = \Delta z^+ = 200; \Delta y_w^+ = 0.8$
 - Compare with Spalart (“*An Approach to wall modeling in large-eddy simulations*”, Physics of Fluids, Vol 12 no. 7, July 2000, pg 1629).

Simulation of Complex, Unsteady Flows Using a Grid-Free Vortex Method

A. Leonard
Graduate Aeronautical Laboratories
California Institute of Technology



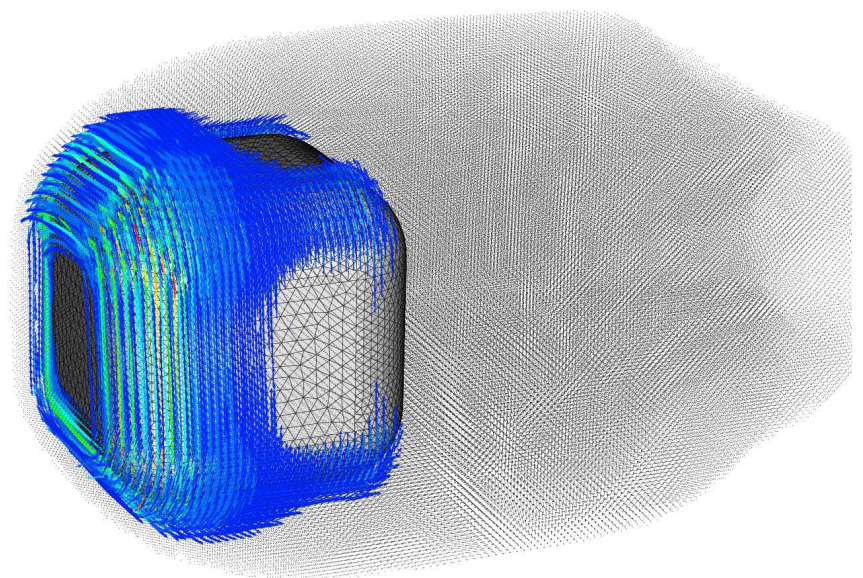
M. Brady, P. Chatelain, M. Rubel

Vortex Code: *Essentials*

- Numerical technique to solve the Navier-Stokes Equations
- Suitable for Direct Simulation and Large-Eddy Simulation
- Uses vorticity (curl of the velocity) as a variable
- Computational elements move with the fluid velocity

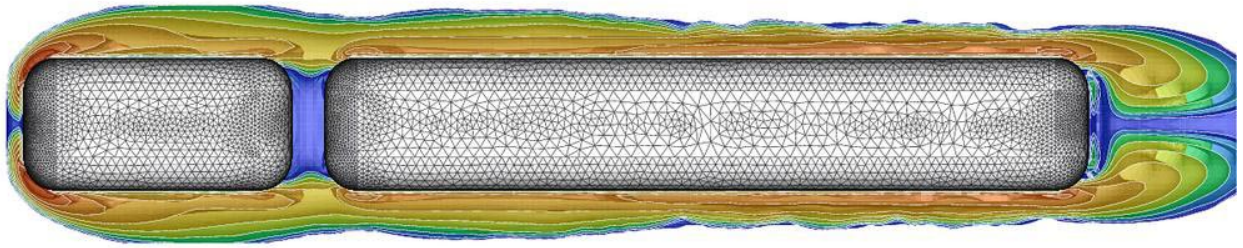
Vortex Code: *Advantages*

- Computational elements only where vorticity is non-zero
- No grid in the flow field
- Only 2D grid on vehicle surface
- Boundary conditions in the far field automatically satisfied



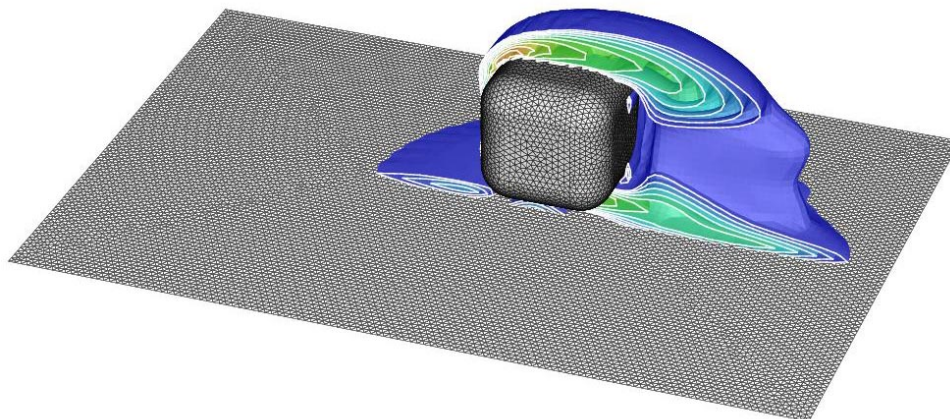
Vortex Code: *Results*

- Tandem rounded prism at GTS-scale (animation)
 $Re = 100$



- Single rounded cube, moderate Reynolds # (animation)
 $Re = 10000$

- Single rounded cube with ground plane (animation)
 $Re = 100$



Vortex Code: *Ongoing and future work*

Long-time simulations with complex bodies require

- Advanced triangulation
- Near-wall treatment
- Adaptively sized triangles
- Ground plane
- Dynamic particle modification in wake

High Reynolds number capability

- DES-like subgrid model
- Advanced subgrid model
- Near-wall vortex elements

Simulation and analysis of truck-like geometries

- With and without ground plane
- Study of leading-edge curvature effects

Large-eddy Simulation: Problem of High Re Wall Turbulence

- Length-scale near wall = $\text{Sqrt}(\text{viscosity}/\text{velocity gradient})$
– wall unit
- Large eddies near wall $O(10 - 100)$ wall units
- LES grids $40 \times 10 \times 10$ wall units near wall
- Pipe flow $\text{Re} = 3 \times 10^{**4} \Rightarrow R_+ = 800$

LES grid volumes = $5 \times 10^{**5}$

Pipe flow $\text{Re} = 3 \times 10^{**7} \Rightarrow R_+ = 550,000$

LES grid volumes = $5 \times 10^{**11}$

grid volumes proportional to $R_+^{**2} = \text{Re_tau}^{**2}$

- Compare competing subgrid models

Wall layer SGS Models

- RANS for attached turbulent boundary layers,
LES for wakes => DES (Spalart et al 1997)

Flow modeling near separation lines questionable

- RANS for near-wall layer
LES for outer turbulent boundary layer plus wakes

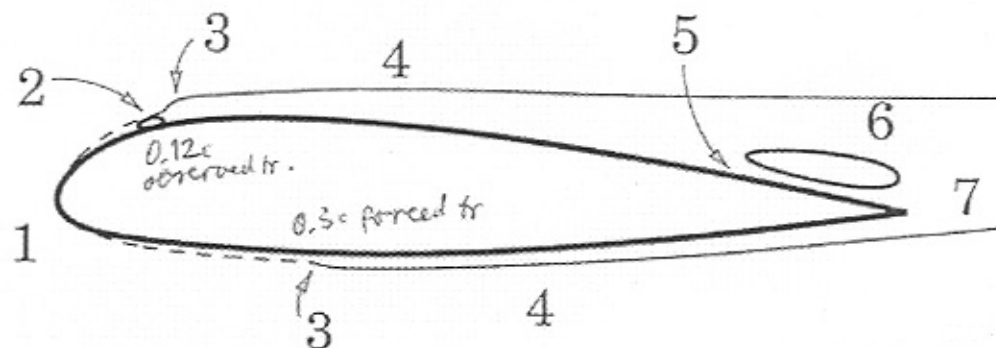
Transition zone between the two a gray area

- Above with stochastic forcing of outer layer
(LES or planetary boundary layers, Mason & Thomson, 1992)

Untested for aero-type turbulent, separating boundary layers

Recent Activity

- LESFOIL: High-lift airfoil at high Reynolds No.
 - Nine European participants
 - Competing subgrid models-LES for outer flow, RANS/wall functions for near-wall
 - Early results not that encouraging
 - Treatment of transition and laminar flow also problems
- LES/RANS Channel Flow
 - Squires, Spalart et al, Phys. Fluids, July 2000
 - Modified Spalart-Allmaras RANS for near-wall
 - Re_{τ} up to 80,000
 - C_f 15% too low



Sketch of the flow regimes around the Aerospatiale A-profile. $Re = 2.1 \cdot 10^6$, $\alpha = 13.3^\circ$.

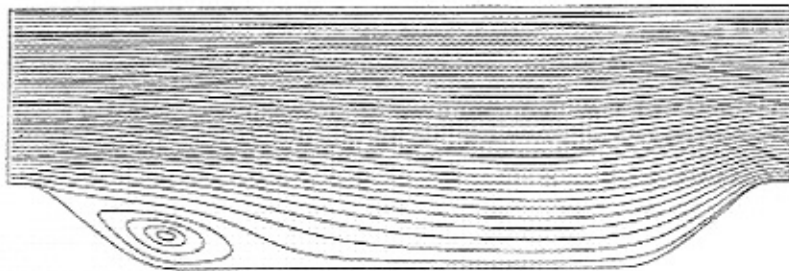
- | | |
|--------------------------|-----------------------------|
| 1 laminar boundary layer | 2 laminar separation bubble |
| 3 transition region | 4 turbulent boundary layer |
| 5 separation point | 6 separation region |
| 7 wake region | |

NACA 4412; Aerospatiale ...

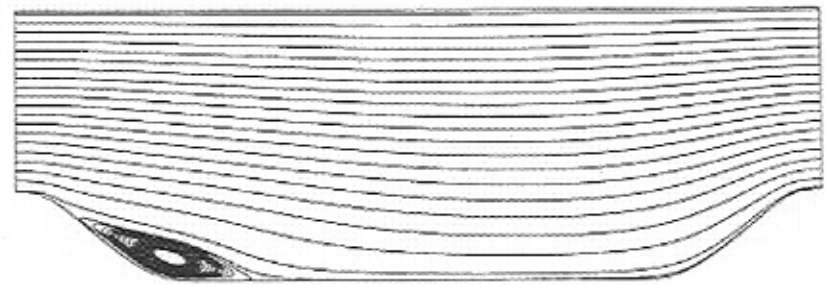
CHALMERS

Lars Davidson, Chalmers University of Technology

- A test case with separation along a curved surface was chosen. Exp. by Heitor & Almeida [1]. Because too high Re-number ($Re_H = 60\,000$) and uncertainty in effect of sidewalls, a modified case was chosen ($Re_H = 12\,000$).



Benchmark LES by Karlsruhe
($202 \times 130 \times 192$)



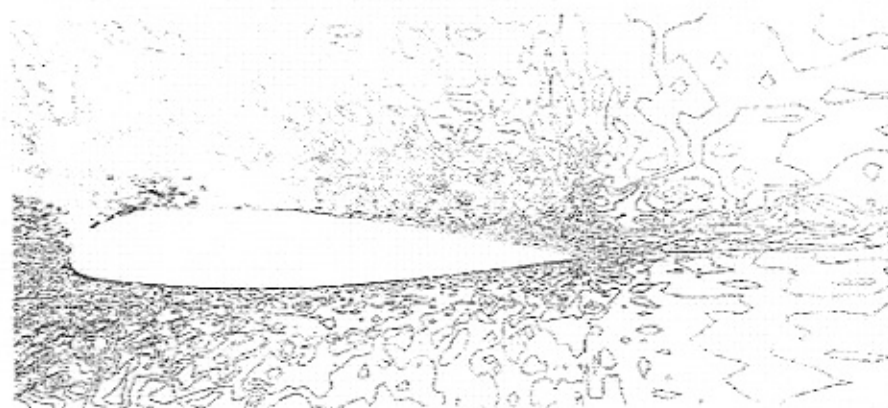
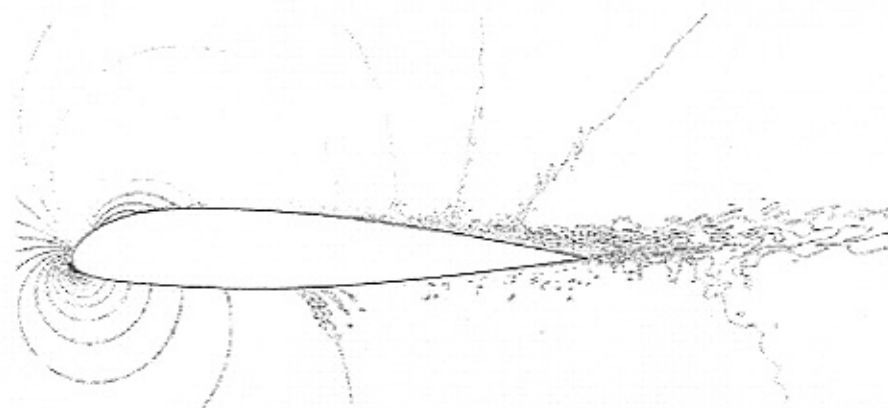
DES by Chalmers ($56 \times 64 \times 48$)

separation small...

CHALMERS

Lars Davidson, Chalmers University of Technology

- Instantaneous \bar{u} velocities. Top: central differencing; bottom: upwind upstream of transition. Results by Chalmers.

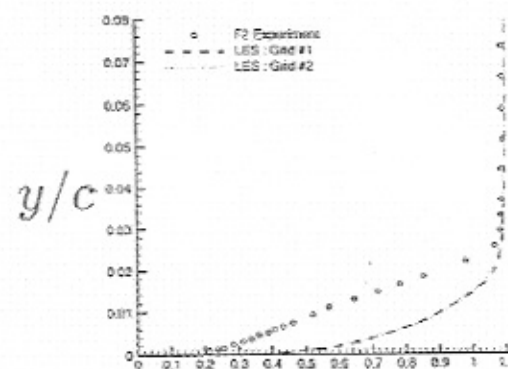
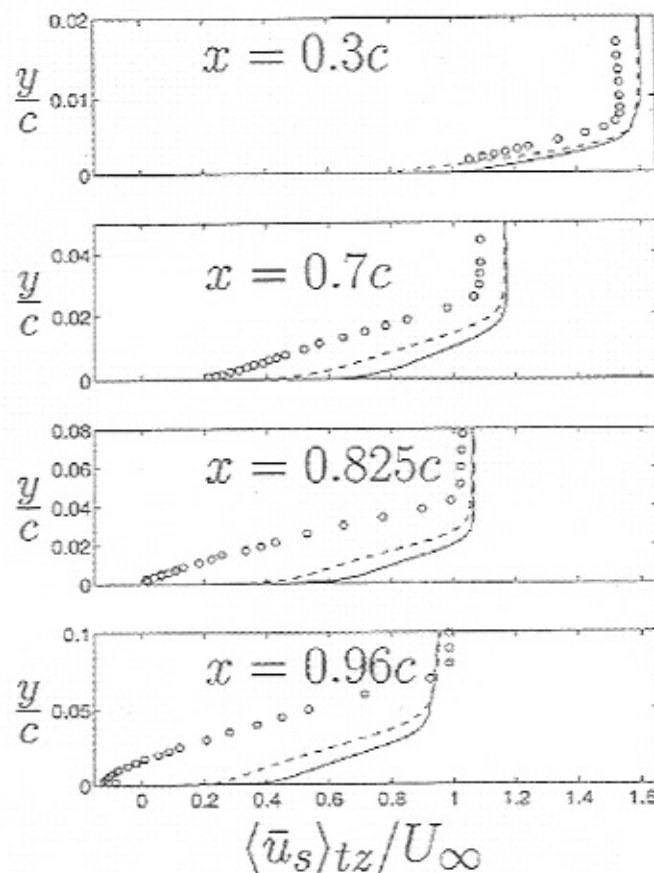
*central**upwind*

CHALMERS

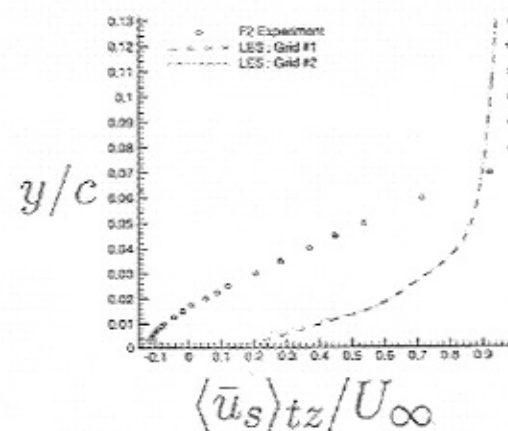
Lars Davidson, Chalmers University of Technology

- Left: Chalmers (dashed=wall functions; solid=no slip);
right: Karlsruhe

mis-predicted transition



$x/c = 0.7$



$x/c = 0.96$

CHALMERS

Lars Davidson, Chalmers University of Technology

DES/LES channel Flow

N. V. Nikitin et al, "An approach to wall modeling in large-eddy simulations,"
Phys. Of Fluids, Vol. 12, No. 7, p. 1629 (2000)

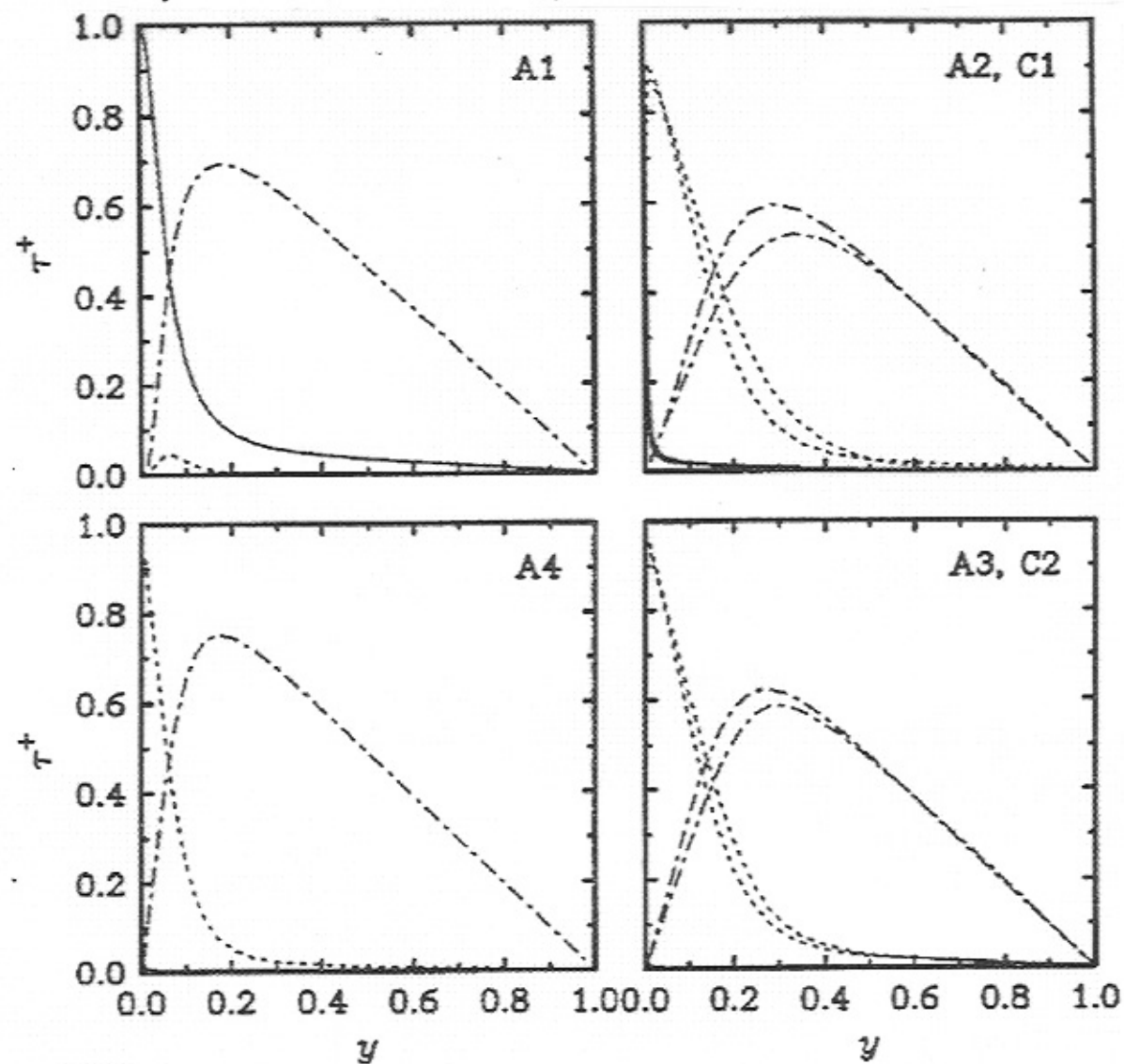


FIG. 1. Shear stresses: viscous (—); modeled (---); resolved (---).

DES/LES channel Flow

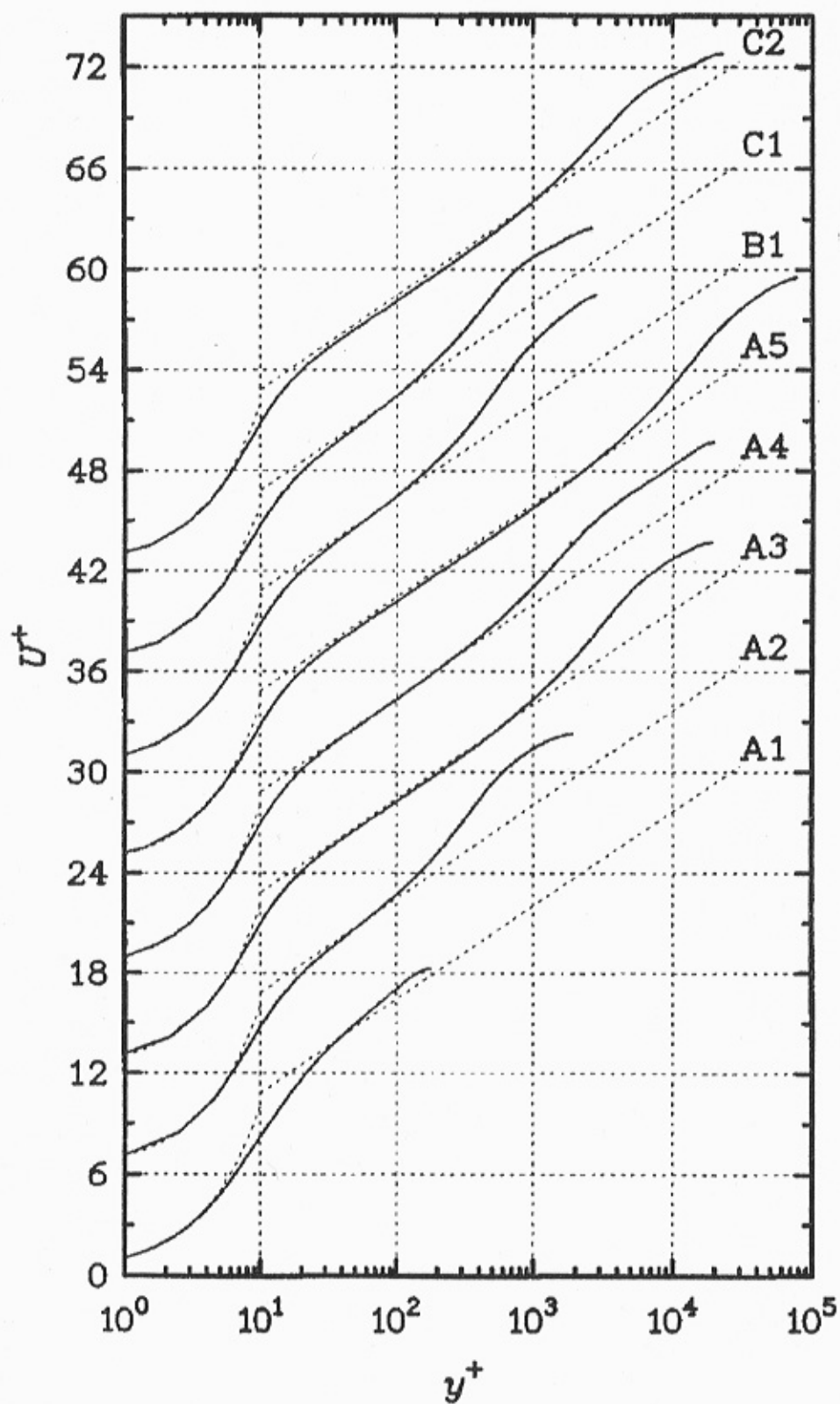


FIG. 2. Velocity: DES (—); $U^+ = y^+$ and $U^+ = \log(y^+)/0.41 + 5.2$ (---).



**Department of Energy
Office of Heavy Vehicle Technologies**

**Near Term Aerodynamic Activities
Report to the AeroDrag Team
August, 2000**

Near Term Aerodynamics: Purpose

- **Focus on aerodynamic activities which can have an impact in the next 1-3 years.**
 - ◆ **Technology test bed / demonstrator**
 - Subject for research studies
 - Platform for testing emerging technologies
 - Platform for working out implementation of known technologies
 - ◆ **CFD design studies**
 - Undercarriage studies (pseudo or partial belly pans)
 - Wheel well studies

Reference Truck

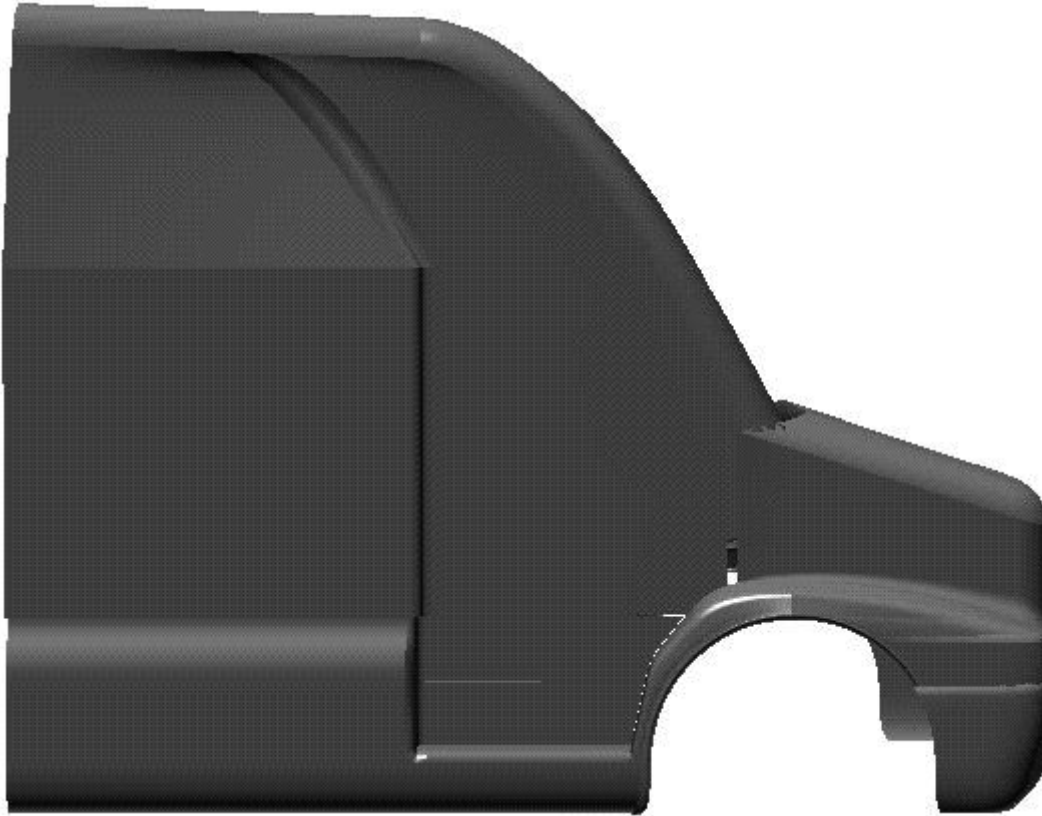
- Outcome of discussions with industry and OHVT participants: **A non-proprietary truck model is needed.**
 - ◆ Provide all members of industry a common baseline to check methods.
 - “What is missing in the world of wind tunnels and aerodynamics is a common baseline”; Luis Novoa, Freightliner.
 - If the government is going to do these tests they better use a model we can all share; Craig Brewster, Peterbilt.
 - ◆ Reference model (that can be published) for CFD work.
 - ◆ Reference model for implementation studies
 - Circulation control, Vortex generators, Boat-tail plates.
 - ◆ Reference for near term design studies

Reference Truck: Development

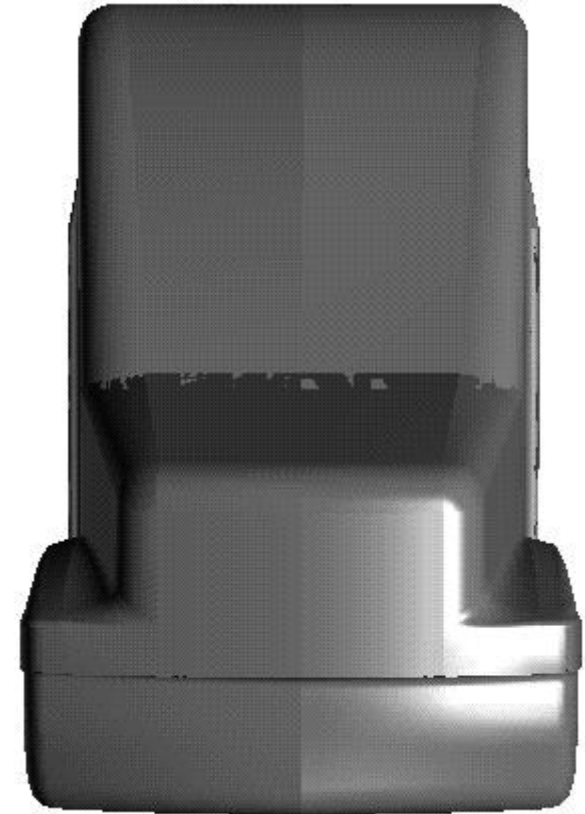
- **Dynacs has begun development of a computer model of a Reference Truck**
 - ◆ **Block out the basic shape**
 - Meet packaging requirements
 - Mimic current design approaches
 - ◆ **Optimize blocked shape**
 - ◆ **Add external aero details (window inset, grill, mirrors, etc.)**
 - ◆ **Add structural and drive train details**
- **Expanded studies will follow**
 - ◆ **New design ideas**
 - ◆ **Detailed aero studies (wind tunnel, CFD, road)**

Reference Truck (Preliminary)

Class 8 Conventional w/ High Rise Sleeper



120 BBC
Axle Forward



18 deg hood
27 deg w/s

75" Sleeper
13'2" Height

Reference Truck (Preliminary)

Class 8 Conventional w/ High Rise Sleeper



120 BBC
Axle Forward

18 deg hood
27 deg w/s

75" Sleeper
13'2" Height

Reference Truck: Extrapolated

- **1st - Computer Model**
 - ◆ CFD studies
 - ◆ Design studies
- **2nd - Sub-scale Wind Tunnel Model**
 - ◆ Reynolds effect studies
 - ◆ Detailed baseline for methods checking
- **3rd - Full-scale Wind Tunnel Model**
- **4th - Full-scale Rolling Mock-up**

Reference Truck: Questions to be addressed

- **Design a good truck or an interesting truck?**
 - ◆ Reynolds number sensitivity designed in
 - ◆ 'A pillar' separation designed in
 - ◆ etc.
- **How much detail should be included?**
 - ◆ Underhood air management
 - ◆ Window insets
 - ◆ Body panel edges
- **How quickly should the design proceed?**

